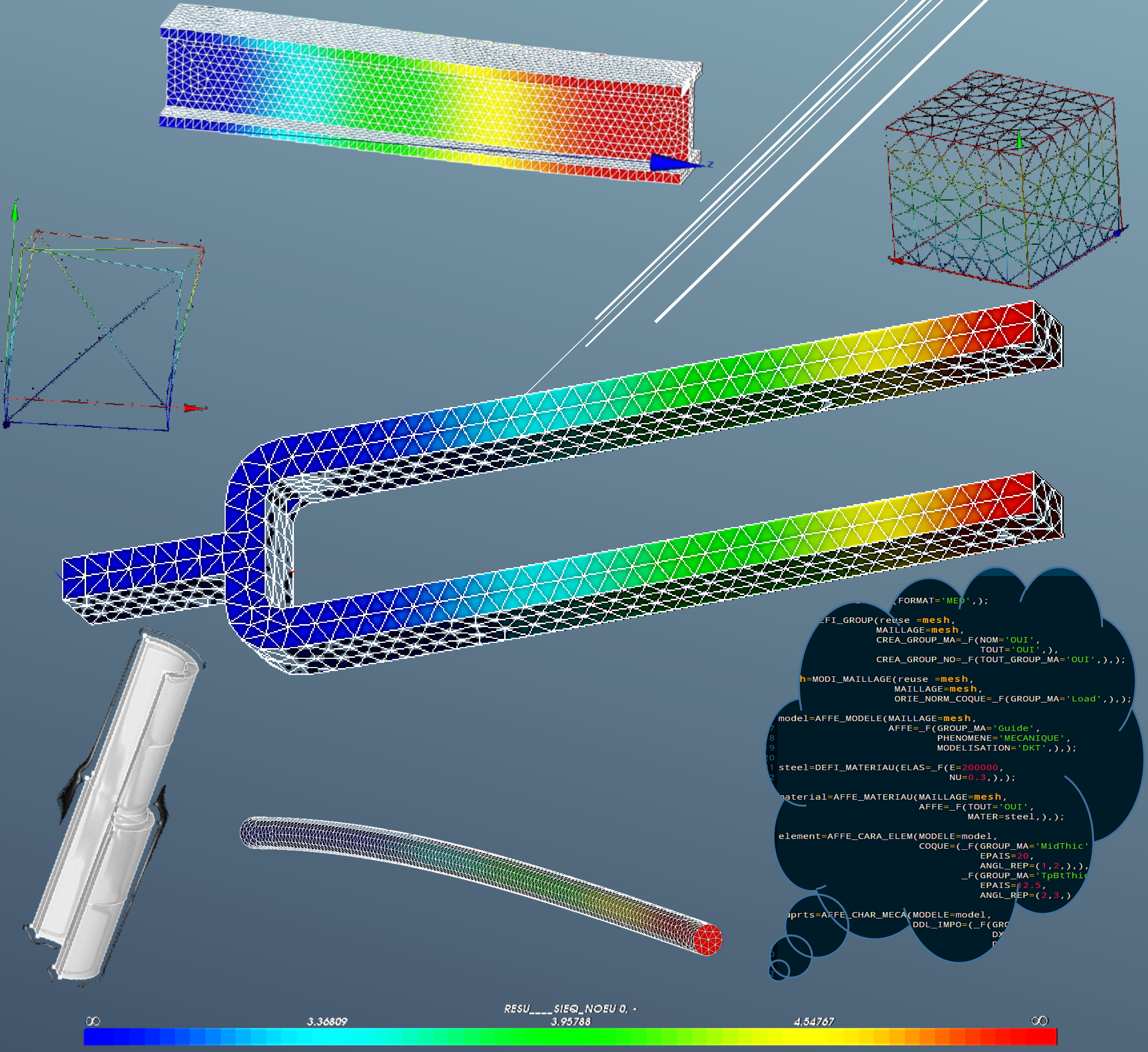
FINITE ELEMENT ANALYSIS

A FIRST COURSE TO

CODE_ASTER

Guide through to Salome_Meca and Code_Aster



```
(FORMAT='MED',);

EFI_GROUP(reuse =mesh,
          MAILLAGE=mesh,
          CREA_GROUP_MA=_F(NOM='OUI',
                           TOUT='OUI',),
          CREA_GROUP_NO=_F(TOUT_GROUP_MA='OUI',),);

h=MODI_MAILLAGE(reuse =mesh,
                MAILLAGE=mesh,
                ORIE_NORM_COQUE=_F(GROUP_MA='Load',),);

model=AFFE_MODELE(MAILLAGE=mesh,
                  AFFE=_F(GROUP_MA='Guide',
                          PHENOMENE='MECANIQUE',
                          MODELISATION='DKT',),);

steel=DEFI_MATERIAU(ELAS=_F(E=200000,
                            NU=0.3,),);

aterial=AFFE_MATERIAU(MAILLAGE=mesh,
                      AFFE=_F(TOUT='OUI',
                              MATER=steel,),);

element=AFFE_CARA_ELEM(MODELE=model,
                       COQUE=(_F(GROUP_MA='MidThic'
                                 EPAIS=20,
                                 ANGL_REP=(1,2,),),
                              _F(GROUP_MA='TpBtThic
                                 EPAIS=12.5,
                                 ANGL_REP=(2,3,)

uprts=AFFE_CHAR_MECA(MODELE=model,
                     DDL_IMPO=(_F(GRO
                                  DX
```

Stefanos Syllignakis
Petr Vosynek

Finite Element Analysis Method using

Open_Source_Software

Step-by-Step Guide…

Finite Element Analysis Method using
Open_Source_Software

Step-by-Step Guide…

Credits and Copyright

Written by: Bc. Syllignakis Stefanos
sylst3f@gmail.com

Main Editor: Ing. Petr Vosynek, Ph.D
petr.vosynek@gmail.com

Review and Editor: Ing. Marek Benešovský
******@vutbr.cz

Preface

The presented material was created within the Erasmus+ project of the student Stefanos Syllignakis under the leadership of Petr Vosynek. It is basically support material for the subject 6KP and its English version 6KP-A (basics of computational modeling using finite element method) taught in the Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology.

Computer labs of 6KP and 6KP-A are composed of active exercises under the current interpretation of the fundamentals associated with the type of elements and also from a separate project for a group of students. The texts were made in two versions, for the computing open_source system Salome_Meca (C_A) and for computing system ANSYS Workbench v16.2.

Contents

Credits and Copyright 3
Preface 3

Introductory 5

General Principles of Code and Platform 5
Presentation of Code_Aster 6
Wide Range of Finite Elements 7
Algorithms and Analysis Methods 8
Creation of a Command File 8
Data Setting and Command Language 9
Salome's Possibilities 10
Code_Aster Possibilities 12
Where to Find Information 16
Useful Links 17
Personal Difficulties During Salome_Meca, Code_Aster Study 17

Chapter_I: Cube 18

Step 1: Purpose of the FE Analysis/ Description of the problem 18
Step 2: Input Values for the FE Analysis 18
Step 3: Model Geometry 19
Step 4: Meshing Geometry 22
Step 5,6,7,8 and 9: Salome_Meca Linear Static Analysis Wizard 24
Step 10: Running the Analysis 26
Step 11: Post Processing the results 27

Chapter_II: Cylinder Rod 32

Step 1: Purpose of the FE Analysis/ Description of the problem 32
Step 2: Input Values for the FE Analysis 32
Step 3: Model Geometry 33
Step 4: Meshing Geometry 35
Step 5,6,7,8 and 9: Salome_Meca Linear Static Analysis Wizard 36
Step 9: Manual editing of command (.comm) file 36
Step 10: Running the Analysis 38
Step 11: Post-Processing of the results 38

Chapter_III: Child Swing 40

Step 1: Purpose of the FE Analysis/ Description of the problem 40
Step 2: Input Values for the FE Analysis 40
Step 3: Model Geometry 41
Step 4: Meshing Geometry 42
Step 5,6,7,8 and 9: Creating command (.comm) file with Eficas 43
Step 10: Run the Analysis 49
Step 11: Post Processing the results 50
Notes 51

Chapter_IV: Beam System 52

Step 1: Purpose of the FE Analysis/ Description of the problem 52
Step 2: Input Values for the FE Analysis 52
Step 3: Model Geometry 53
Step 4: Meshing Geometry 55
Step 5,6,7,8 and 9: Creating the command (.comm) file 56
Step 10: Run the Analysis 58
Step 11: Post Processing the results 58

Chapter_V: Plate 60

Step 1: Purpose of the FE Analysis/ Description of the problem 61
Step 2: Input Values for the FE Analysis 61
Step 3: Model Geometry 62
Step 4: Meshing the Geometry 64
Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis 65
Step 11: Post Processing the Results 70

Modifying the Geometry 71

Step I: Model Geometry 71
Step II: Meshing Geometry 73
Step III: Creating the command file, run the Analysis 74
Step IV: Post Processing the results 75

Finite Element Convergence 76

Plate Bar: 76
Results 77
Filleted_Plate_Bar: 79
Results 80

Chapter_VI: Shaft 81

Step 1: Purpose of the FE Analysis/ Description of the problem 81

Examples before starting our model 82

Step 2: Input Values for the FE Analysis 83
Step 3: Model Geometry 84
Step 4: Meshing of Geometry 86
Step 5, 6, 7, 8, 9, 10: Salome_Meca Linear Static Analysis Wizard, Running the Analysis 87
Step 11: Post Processing of the Results 88
Stress Concentration Factor 89

Hand Calculations VS Computational Calculations of Stress Concentration 90

Hand Calculations 90
Computational Calculations 90
Solving the Equation 90

Redefining Mesh 91

Chapter_VI: Level of Geometry 92

Step 1: Purpose of the FE Analysis/ Description of the Problem 93
Step 2: Input Values for the FE Analysis 94

i. Beam_Elements 95

Step 3: Model Geometry 95
Step 4: Meshing Geometry 96
Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis 97

Step $: Creating a Macro Command 99

Step 11: Post Processing the Results 104

ii. Solid_Elements 105

Step 3: Model Geometry 105
Step 4: Meshing Geometry 107
Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis 108
Step 11: Post Processing the Results 110

iii. Surface_Elements 111

Step 3: Model Geometry 111
Step 4: Meshing Geometry 113

Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis 114
Step 11: Post Processing the Results 116

Chapter_VIII: Tunning Fork 117

Step 1: Purpose of the FE Analysis/ Description of the Problem 118
Step 2: Input Values for the FE Analysis 118
Step 3: Model Geometry 119
Step 4: Meshing Geometry 123
Step 5, 6, 7, 8, 9, 10: Salome_Meca Modal Analysis Wizard 124
Step 11: Post Processing of the Results 127

Step $: Modify Model 128

i. Changing the Material 128
ii. Changing the Dimensions. 129

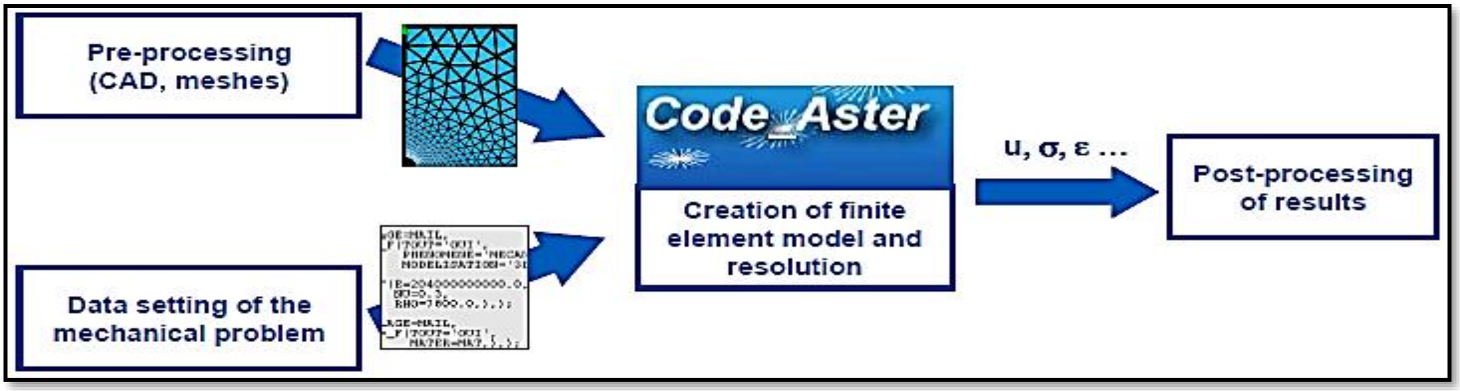
Introductory

Reference --> EDF, *Code_Aster and Salome_Meca course material.*

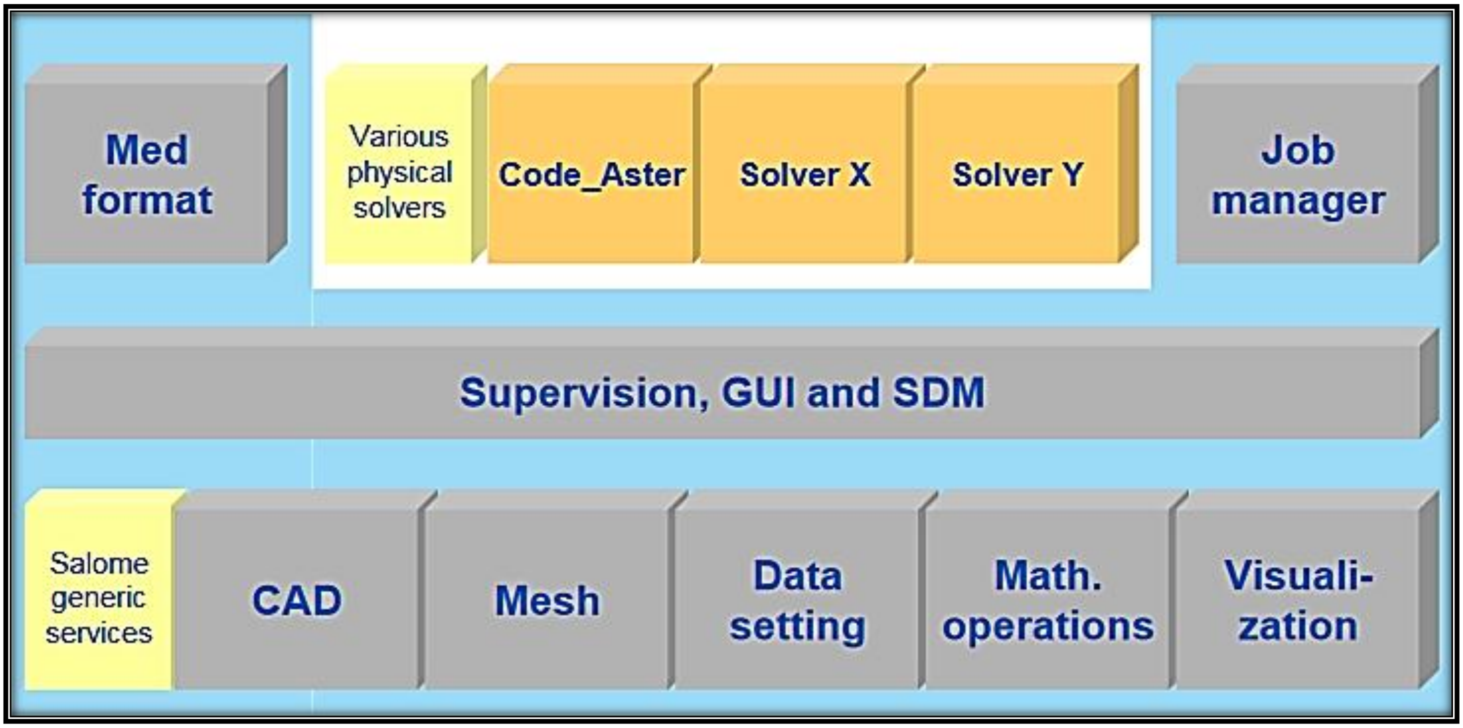
## General Principles of Code and Platform

# General Principles of Code and Platform:

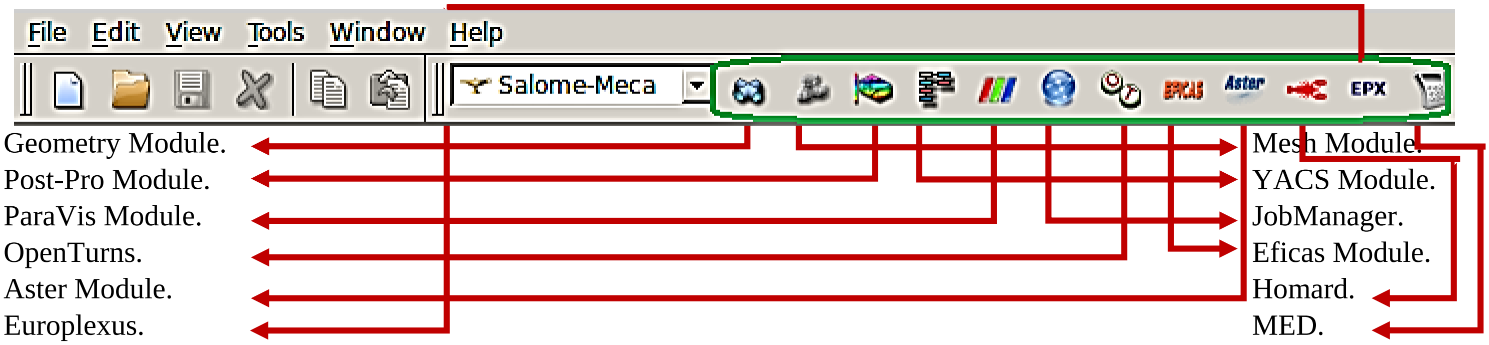
- Code_Aster is a “stand-alone” thermos-mechanical solver.
  - No integrated GUI to create geometries and meshes.
  - No colorful post-processing.
  - With study data prepared in a text file.
  - Input: Mesh and data setting.
  - Output: Physical fields (displacement, stress-strain, temperature).


Pre-processing (CAD, meshes)
Data setting of the mechanical problem
Code_Aster
Creation of finite element model and resolution
u, σ, ε ...
Post-processing of results

# Salome is a generic framework for pre- and post-processing.



- Code_Aster is a thermo-mechanical solver.
- Salome is a generic platform for physical simulations.
  - Salome_Meca = Salome + Code_Aster (Software integration).
- Advantages:
  - Easy installation of a complete framework (Linux).
  - A consistent and continuous graphical environment.
    - Access from different modules to main Salome study elements: meshes, results.
    - Graphical selection of topological entities for data setting of Code_Aster.
  - Possibility of using different pre- and post-processing tools.
    - Importation of meshes and geometries prepared by GEOM and SMESH Salome modules.
    - Importation of different input mesh formats and output result formats.
  - Possibility for a "stand-alone" use of Code_Aster solver.
- Salome_Meca is the integration of Code_Aster solver in the Salome Platform.



Presentation of Code_Aster

- Code_Aster is an acronym for analysis of structured and thermos mechanics for studies and research, is a general Finite Element Analysis software coming from EDF (Electricite De France) R&D Department.
- It can handle mechanical thermal and associated phenome in all sort of analysis: Linear Statics, Non-Linear Statics, Dynamics, Thermics and more.
- Due to its numerous capabilities, Code_Aster is a very complex affair and its somewhat unfriendly user interface makes the learning curve quite steep at the beginning.
- An all-purpose code for thermos-mechanical study of structures.
  - With a wide variety of models:
    - More than 400 finite elements: 3D, 2D, Shells, Beams, Pipes.
    - More than 100 constitutive laws.
    - A wide range of solvers: Mechanical Statics and Dynamics, Vibrations, Modal and Harmonic Analysis, Thermo-Hydro-Mechanical Coupled Problems, Thermic, Acoustics, etc.
  - A computational software used by engineers, experts and researchers.
    - Studies: A need of robust, reliable, tested and qualified industrial simulation code at EDF.
    - Researches: Continuous integration of new models in the development versions.
- Solving three types of non-linear problems.
  - Material Behavior: About a hundred nonlinear constitutive laws.
  - Kinematics: Large displacements, large strains, large rotations.
  - Contact and/ or friction.
- Advanced features in mechanics.
  - Porous media, fracture mechanics, fatigue, damage, metallurgy, seismic analysis, rotating systems.

- Code_Aster Definitions.
    - FONCTIONS [FR]: Tabulated (discrete) function depending on one parameter.
    - NAPPE [FR]: Tabulated (discrete) function depending on two parameters.
    - FORMULE [FR]: Continuum formula depending on several parameters.
- Code_Aster Parameters:
    - ABSC_CURV → Curvilinear abscissa.
    - DX, DY, DZ → Displacements along X, Y, Z.
    - DRX, DRY, DRZ → Rotation along X, Y, Z.
    - X, Y, Z → Coordinates X, Y, Z.
    - EPSI → Strain.
    - SIGM → Stress.
    - INST → Time.
    - TEMP → Temperature.
- File Extensions:
    - .comm → Command File.
    - .mmed → MED file containing mesh.
    - .mess → Text file containing output of the solver (message, errors, warnings, etc.).
    - .resu → Text file containing results of the simulation in a table format.
    - .rmed → MED file containing results of the simulation.
    - .base → Folder containing the output database of the solver.
    - .bhdf → File containing the base of calculations in HDF format.
    - .btc → Launch script generated by the service.
- Eficas standard concepts:
    - DEBUT [Start].
    - Material definition.
    - What type of Mesh to read.
    - Type of modeling [1D, 2D, 3D, mechanical, thermal].
    - Assign previously defined material to the model.
    - Add geometric boundary conditions.
    - Add load boundary conditions.
    - Define the type of analysis [static, linear, non-linear, dynamic, etc.].
    - Results to be calculated at the elements and nodes.
    - Save the results to a MED file.
    - FIN [Finish].

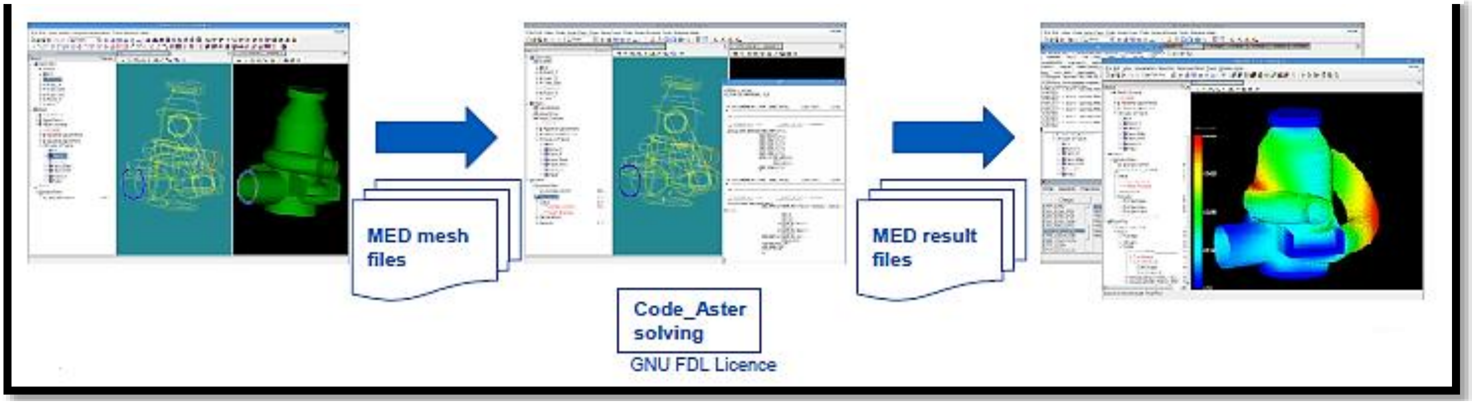
Wide Range of Finite Elements

- Continuous mechanics.
    - 3D --> Linear, quadratic, reduced or full integration.
    - 2D --> Plane strain, plane stress, axi-symmetry.
      --> Integration of non-linear behavior in plane stress.
- Structural elements.
    - 2D elements: Shells, plates…
    - 1D elements: Beams, bars, cables, pipes…
- Connections and assemblies.
    - Linear relationships between degrees of freedom, transmission of torques effort.
- Discontinuous media (cracks and joints).
    - XFEM level-sets.
    - Joint elements and CZM (Cohesive Zone Model).

Algorithms and Analysis Methods

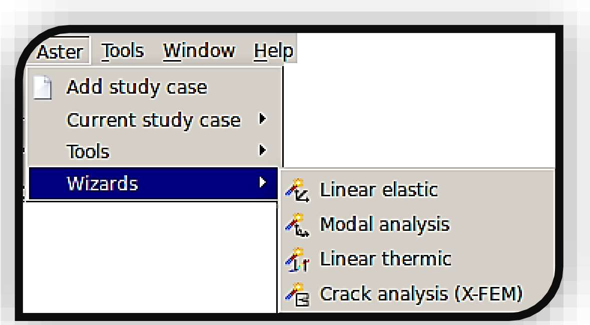
- Mechanical solvers.
  - Linear or non-linear static: **MECA_STATIQUE, STAT_NON_LINE…**
  - Dynamic on physical basis: **DYNA_LINE_TRAN, DYNA_NON_LINE…**
  - Modal analysis: **CALC_MODAL, MODE_ITER_*…**
  - Dynamic on modal basis: **DYNA_TRAN_MODAL, DYNA_VIBRA…**
- Other physics.
  - Thermic: **THER_LINEAIRE, THER_NON_LINE…**
  - Acoustics: **PHENOMENE ACOUSTIQUE…**
  - Metallurgy (for welding applications).
  - FSI: Fluid structure interaction.
  - Thermos-hydro-mechanical coupling.
- Tools for resolution.
  - Sub structuring, control of the nonlinear algorithms.
  - Several algebraic solvers, sequential or parallel, direct or iterative.
  - Post-processing tools: **CALC_CHAMP, POST_CHAMP, POST_DYNA_*…**

Sketching Geometry → Arranging Mesh Inputs → Visualizing Mesh Outcome → Adjusting Code_Aster → C_Aster Solver → Post-Processing Visual.

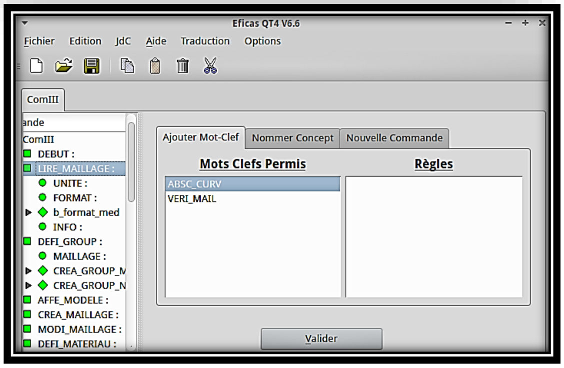


CAD Modeler/ Salome (GEOM) → Mesh tool/ Salome (SMESH) → Data setting/ Eficas, Wizard → Computation / ASTK → Visualization App/ PARAVIS

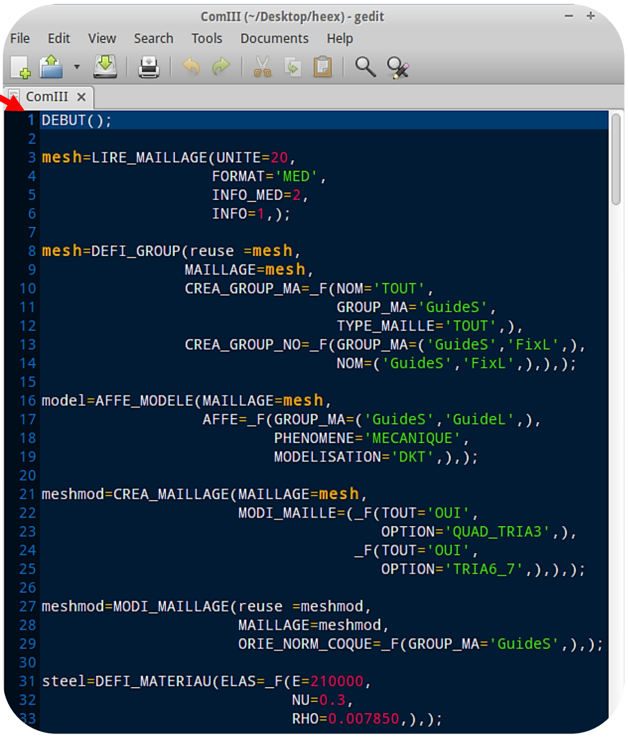
Creation of a Command File



- With a Wizard.
  - Available in Salome_Meca only.
  - 4 different wizards in Salome_Meca 2013.1.
    - Linear Elastic Analysis.
    - Modal Analysis.
    - Linear Thermal Analysis.
    - Crack Analysis.
- With the graphical command file editor (Eficas App).
  - Provided with Salome_Meca.
  - Provided with Code_Aster standalone.
  - Cannot handle python control flow instructions (if, while, for).



# With your favorite text editor (ex. Gedit,..).



ComIII (~/Desktop/heex) - gedit
File Edit View Search Tools Documents Help
ComIII

```
DEBUT();

mesh=LIRE_MAILLAGE(UNITE=20,
                   FORMAT='MED',
                   INFO_MED=2,
                   INFO=1,);

mesh=DEFI_GROUP(reuse =mesh,
                MAILLAGE=mesh,
                CREA_GROUP_MA=_F(NOM='TOUT',
                                 GROUP_MA='GuideS',
                                 TYPE_MAILLE='TOUT',),
                CREA_GROUP_NO=_F(GROUP_MA=('GuideS','FixL',),
                                 NOM=('GuideS','FixL',),),);

model=AFFE_MODELE(MAILLAGE=mesh,
                  AFFE=_F(GROUP_MA=('GuideS','GuideL',),
                          PHENOMENE='MECANIQUE',
                          MODELISATION='DKT',),);

meshmod=CREA_MAILLAGE(MAILLAGE=mesh,
                      MODI_MAILLE=(_F(TOUT='OUI',
                                      OPTION='QUAD_TRIA3',),
                                   _F(TOUT='OUI',
                                      OPTION='TRIA6_7',),),);

meshmod=MODI_MAILLAGE(reuse =meshmod,
                      MAILLAGE=meshmod,
                      ORIE_NORM_COQUE=_F(GROUP_MA='GuideS',),);

steel=DEFI_MATERIAU(ELAS=_F(E=210000,
                            NU=0.3,
                            RHO=0.007850,),);
```

Data Setting and Command Language

# Command file is also a Python script.
  - However, we should only focus on the Code_Aster commands.

# Command file composes of a sequence of Code_Aster specific commands.
  - Each specific command composes of keywords and defines, assigns or uses data as input.

# Most of the commands produce “concepts”.
  - On the left side of the equal sign (=).
  - The concepts generated by one command can be used as an input to the following command.

# Command file contains no geometry description.

# Examples:
  - steel = DEFI_MATERIAU (ELAS=_F(E=210000, NU=0.3, RHO=0.007850,) ,);

Command

Argument of the Simple Keyword

Factor Keyword

Name of the concept, User choice

Simple Keyword

  - IMPR_RESU (RESU=_F (RESULTAT = RESU),);

Factor Keyword

Command

Argument of the Simple Keyword

No Concept is Produced

Simple Keyword

  - “reuse” keyword is used to extend an existing concept.

mesh = LIRE_MAILLAGE (FORMAT=’MED’)
mesh = DEFI_GROUP (reuse = mesh,
MAILLAGE = mesh,
CREA_GROUP_NO = _F (GROUP_MA = ‘upper’,) ,);

Reuse of the MA Concept Defined by LIRE_MAILLAGE

Indicator of a Factor Keyword

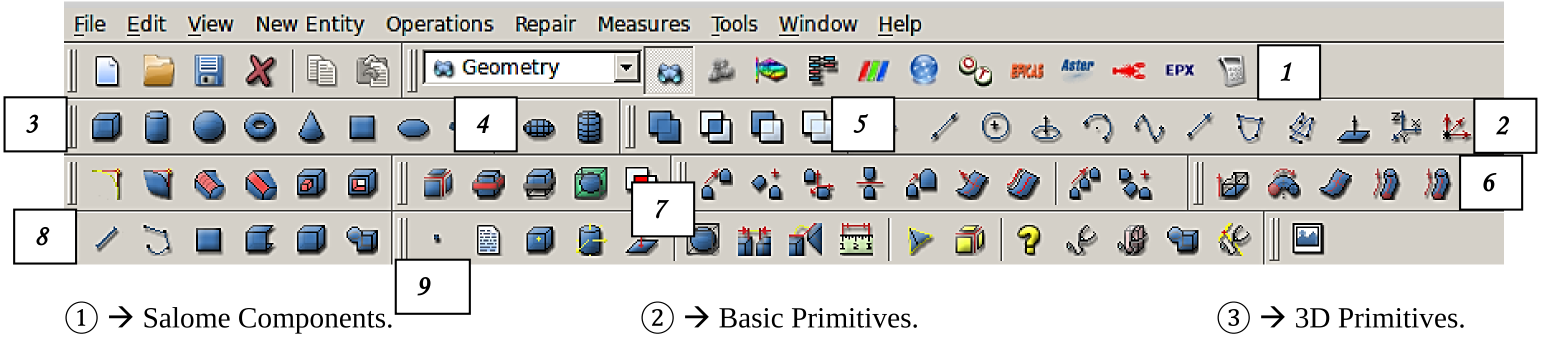
  - Since commands are sequential, a concept must be created before being used.

MESH = LIRE_MAILLAGE ();
STEEL = DEFI_MATERIAU (ELAS = _F (E=20500e06, NU=0.3,) ,);
CHAMT = AFFE_MATERIAU (MAILLAGE=**MESH**,
AFFE = _F (TOUT=’OUI’,
MATER=**STEEL**,) ,);

Salome's Possibilities

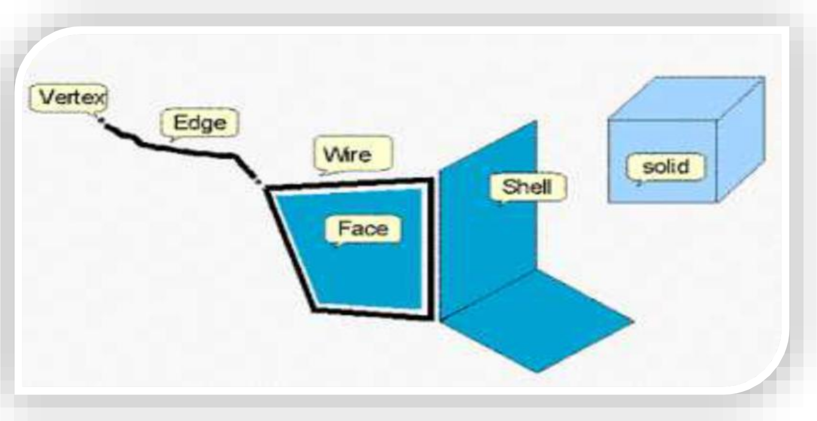
Reference --> EDF, *Code_Aster and Salome_Meca course material.*

- What can we do with Salome:
  - Import and export, repair and clean, create and modify geometries (CAD).
  - Mesh, quality control, import/ export meshes.
  - Manipulate the physical and numerical properties of a geometry.
  - Manage the various stages of a computation: receive data, setting up a calculation, return results.
  - Run the computation sequences and coupling between solvers.
  - View and post-process the results.
- Salome, GEOM Module:
  - Design of geometric objects.
  - Import/ export of objects of different CAD formats.
  - Repairs and correction of CAD models.
  - Adaptation of CAD models for computer simulation.
  - Based on the Open Cascade technology.
- Salome, GEOM Module: Import & Viewers
  - Various import/ export formats: STEP, IGES, BREP, STL, ACIS.
  - Graphic functionalities: Transparency, coloring, shading/ wireframe, rotation, zoom, pan.
- Salome, GEOM Module: GUI



① → Salome Components.
② → Basic Primitives.
③ → 3D Primitives.
④ → Advanced Primitives.
⑤ → Boolean Operations.
⑥ → Generations.
⑦ → Transformations.
⑧ → Constructions.
⑨ → Measures.

- Salome, GEOM Module: Terminology
  - Vertex.
  - Edge.
  - Wire (set of edges).
  - Face (constructed from a closed wire).
  - Shell (set of faces).
  - Solid (constructed with a second shell).
  - Compound (set of any types).



- Salome, GEOM Module: Conception
  - Geometric Primitives
    - 1D: Line, circle, ellipse, arc, curve, vector.
    - 2D: Plan, working plan.
    - 3D: Parallelepiped, cylinder, sphere, torus, cone.
    - Sketch 2D/3D: Construction of complex lines or surfaces.
    - Construction of elementary objects: Vertex, edge, wire, shell, solid, compound.
    - Advanced Primitives: Tee pipe.

- Operations
  - Boolean: Merge, join, cut, intersect.
  - Fillet, chamfer, partition.
  - Explode: decomposition into basic objects.
- Generations
  - Extrusion, revolution, filling, pipes.
- Transformations
  - Translate, rotate, mirror, scale, offset.
  - Multi-translation, multi-rotation.

# Salome, GEOM Module: Partitions and Objects

- Define the relevant topological entities to ease mesh creation, calculation setup and post-processing result.
- Geometric Groups: Manage geometric objects from which can be created element/ node groups required when assigning boundary conditions or material properties.
- Geometric Operations to Define Partitions in a Mesh: Non-manifold geometry, for hexahedral meshing splitting non-hexahedral shape into hexahedral ones, define different refinements, merge.

# Salome, MESH Module

- The geometric model is imported from GEOM Module.
- Information and quality control of meshes.
- Groups of nodes or elements and operations on these groups.
- Various import/ export formats: MED, UNV, STL, CHNS.
- Modifications of meshes
- Principles:
  - One algorithm for each dimension [1D: Wire Discretization].
  - One hypothesis for each algorithm [Nb. Segments].
- The mesh calculation starts from the smallest to the largest dimension:
  - 0D → 1D → 2D → 3D.

# Salome, SMESH Module: Algorithms

- For each dimension, several algorithms are available:
  - 1D: Wire Discretization, Projection 1D…
  - 2D: Quadrangle (mapping), NETGEN 1D-2D, BLSurf…
  - 3D: Tetrahedrons (NETGEN), Tetrahedrons (GHS3D), Hexahedrons (i, j, k)…
- Some algorithms are multidimensional:
  - 1D-2D: BLSurf, NETGEN 1D-2D…
  - 1D-2D-3D: NETGEN 1D-2D-3D…

# Salome, SMESH Module: Hypothesis

- The parameters of a meshing algorithm are set through a Hypothesis.
- Ex. for 1D mesh:
  - Algorithm 1D: Wire Discretization.
  - Associated Hypothesis: Nb. Segments = 4, Equidistant Distribution.
- Ex. for 2D mesh:
  - Algorithm 1D: Wire Discretization.
  - Associated Hypothesis: Nb. Segments = 4, Equidistant Distribution
  - Algorithm 2D: Quadrangle (mapping).

# Salome, PARAVIS Module: Terminology

- The Salome post-processing module based on ParaView.
  - With functionalities added by EDF.
    - MED interface.
    - Integration points.
    - Modal animation.

- Filters.
  - In ParaViS, the data is managed by means of filters.
    - Ex. Deformed Shape, cutting plane …
  - The filters depend on the type of data. They can be chained.
    - Initial data → Filters 1 → Filter 2 → …
- Views.
  - A surface on which you can see the data.
    - Ex. 3D, 2D, histogram, plot…
- Displays.
  - The data can be seen in different ways in different views.
    - Ex. Surface, Wireframe, PointSprite…

Code_Aster Possibilities

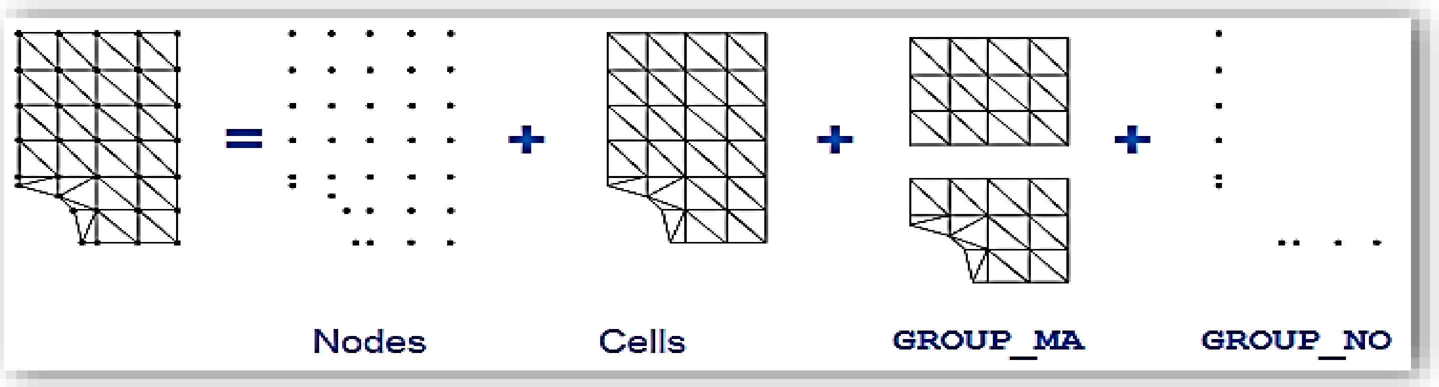
# Special commands: DEBUT, FIN, POURSUITE

- The DEBUT command:
  - Begins execution, ignoring the previous lines.
  - Sets the logical units to be used for messages and files results.
  - Basic usage: **DEBUT ( )**
- The POURSUITE command:
  - Restarts execution from a base provided as an input.
  - Useful to continue a calculation initiated with the same version of Code_Aster.
  - Recommended to decouple the calculation of post-treatment.
  - Basic usage: Calculation: **DEBUT ( )** **…** **FIN ( )**

    Post: **POURSUITE ( ) …** **FIN ( )**
- The FIN command:
  - Ends the command file and ends the run, ignoring the following lines.
  - Closes the base at the end of execution: folder containing all the concepts generated during the calculation (mesh data, intermediate structures, results).
  - Specifies the format used for the produced base: .hdf or Aster.

# What is a Mesh?

- Coordinates of nodes.
- Cells defined by their connectivity.
- Groups of cells (GROUP_MA) and groups of nodes (GROUP_NO).



# Read the Mesh

- Meshes at Aster format or MED formats:
  - Code_Aster format:
    - **mymesh = LIRE_MAILLAGE ( )**
  - MED format:
    - **mymesh = LIRE_MAILLAGE (FORMAT = ‘MED’)**
- Meshes of other formats:
  - Commands PRE_***: PRE_GIBI, PRE_IDEAS, PRE_GMSH.
  - Ex. **PRE_*** ( )**

    mymesh = LIRE_MAILLAGE ( )
- Results:
  - Command **IMPR_RESU**
  - For instance in MED format: **IMPR_RESU (FORMAT = ‘MED’,**

    RESU = _F (RESULTAT = myresult,) ,);

# The Choice of Finite Elements

- A finite element is:
  - A geometric description, provided by the mesh.
  - Shape functions.
  - Degrees of freedom.
- Its choice determines:
  - The equations that are solved.
  - Discretization and integration hypothesis.
  - The output fields.
- 3D elements examples:
  - **MODELISATION = ‘3D’**
    ‘3D_SI’
    ‘3D_INCO’
- 2D element examples (plane strain, plane stress, axi-symmetry):
  - **MODELISATION = ‘D_PLAN’**
    ‘C_PLAN’
    ‘AXIS’
- 1D element examples (beams, pipes, cables):
  - **MODELISATION = ‘POU_D_T’**
    ‘TUYAU’
- Shell elements examples (plane strain, plane stress, axi-symmetry):
  - **MODELISATION = ‘DKT’**
    ‘DST’
    ‘COQUE_3D’

But also many other mechanical 3D, 2D, 1D elements like: Sub-integrated, incompressible, non-local, fluid-structure coupled THM….

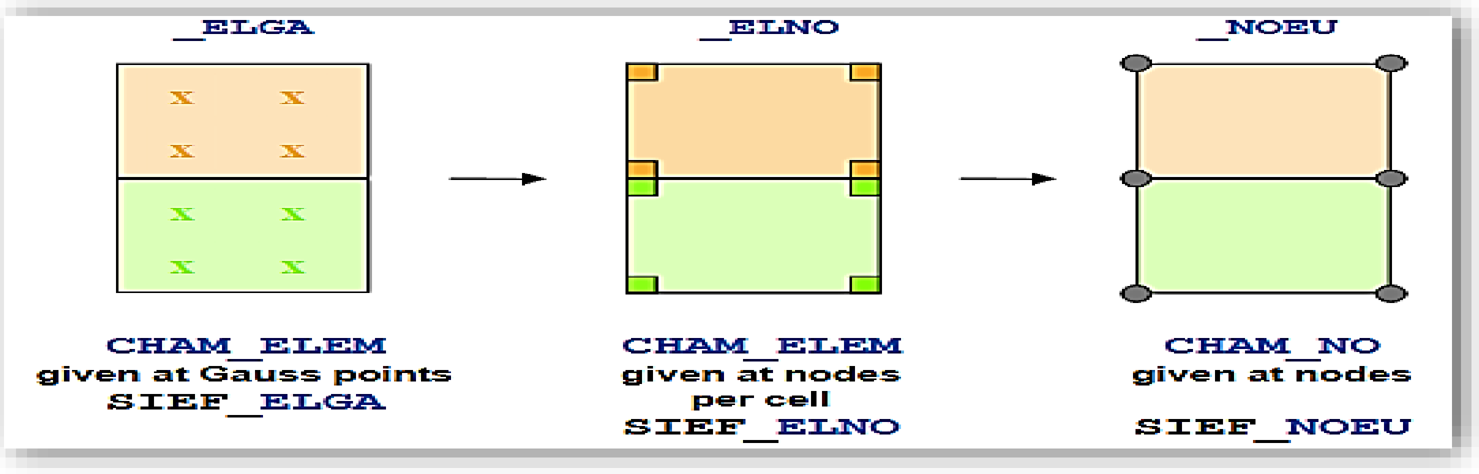
- The Definition and Assignment of Materials:
    - A material is the definition of numerical parameters
        - Units of measurement are not managed by Code_Aster, the user must use a consistent system of units between materials, unit length of the mesh, loading data, like:
            - Coordinates of the mesh nodes: mm → m.
            - Young's modulus: MPa → Pa.
            - Applied force: N → N.
            - Stress obtained as a result: MPa → Pa.
    - The material must be assigned.
        - Assignment to a geometrical area: Groups of finite elements designated by the name of groups of cells.
    - The material must be consistent with the assumptions of resolution (constitutive law).
    - Be careful to assign what has been defined, then use what has been assigned.
- The Characteristics of Structural Elements:
    - Shells, plates, beam, pipes, discrete elements…
    - Geometric information not provided by the mesh:
        - Shells: Thickness, reference direction in the tangent plane.
        - Beams: Cross-section, orientation of the principal axes of inertia around the neutral axis, curvature of the curved elements.
        - Discrete Elements (spring, mass/ inertia, damping): Values of the stiffness, mass or damping matrices.
        - Bars or Cables: Area of Cross-section.
        - 3D or 2D Continuous Media Elements: Local axes defined with respect to the anisotropy directions.
- Loadings:
    - Commands **AFFE_CHAR_*****
    - Loadings inside the continuous media: **PESANTEUR, ROTATION, FORCE_NODALE**…
    - Edge loadings: **FORCE_FACE, FORCE_ARETE, PRES_REP**…
    - Loadings specific to structural elements: **FORCE_POUTRE, FORCE_COQUE, FORCE_TUYAU**…
    - Pre-established relationships on the degrees of freedom:
        - **DDL_IMPO**: DoF imposed on a node or group.
        - **FACE_IMPO**: DoF imposed on the nodes of a cell or a group of cells.
        - **LIAISON_SOLIDE**: Regasification of a set of nodes or cells.
    - Be careful to the constistency with the degrees of freedom allowed by the chosen finite element.
- Solving the Problem:
    - There are about fifteen resolution operators to solve the physical problems:
        - Thermic: **THER_LINEAIRE, THER_NON_LINE**
        - Mechanics: **MECA_STATIQUE, STAT_NON_LINE**
        - Dynamics: **DYNA_VIBRA, DYNA_NON_LINE**
        - Modal Calculation: **MODE_ITER_SIMULT, MODE_ITER_INV**
    - One must give the description of the problem set prior solving.
        - The model: **MODELE**
        - Materials: **CHAM_MATER**
        - Geometrical Characteristics: **CARA_ELEM**
        - Loading: **EXCIT**
        - A time list if necessary: **LIST_INST**
    - One can also change settings on the resolution algorithm.
        - In most cases, the default values are suitable.

# Choosing the Linear Solver.

- The solver can be chosen via the keyword **SOLVEUR/ METHODE**.
- Relevant depending on the problem to be solved.
- Direct solvers:
  - MULT_FRONT (multi-frontal): Default method. Universal solver, very robust. Not recommended for mixed models of X-FEM, incompressible…
  - MUMPS (multi-frontal massively parallel sparse direct solver): External solver. Same scope as MULT_FRONT. Provides access to parallelism.
- Iterative solvers:
  - GCPC (preconditioned conjugate gradient): Recommended method for thermic. Useful for well-conditioned, large problems.
  - PETSC: External multi-method solver. Very robust when associated with pre-conditioner.

# Fields of Quantity.

- Location of the values:
  - Fields on nodes: **NOEU**;
  - On the elements:
    - Field by element on Gauss Points: **ELGA**;
    - Field by element on Nodes: **ELNO**;
    - Constant field on element: **ELEM**;
- Naming rules **XXXX_YYYY**:
  - 4 first letters: name of the quantity, **SIGM**, **EPSI**, **ERRE**;
  - 4 middle letters: location, **NOEU**, **ELGA**, **ELNO**, **ELEM**;
- Location of the values:
  - Three locations:



# Concept Resultat.

- Resolution operators produce typed data structures: resultat.
- The type depends on the operator: **EVOL_ELAS, EVOL_NOLI, EVOL_THER, MODE_MECA**…
- At each computation step, one or more fields are stored in the data structure: resultat.
- Fields are identified by access variables: **INST, NUME_ORDRE, FREQ, NUME_MODE**…

- Calculate Physical Fields or Perform Mathematical Operation: **CALC_CHAMP /POST_ELEM**.
  - Produce fields and tables.
  - **CALC_CHAMP**
    - Create or complete a DS resultat.
    - Calculation of stresses, strains, energies, criteria, error indicators…
  - **POST_ELEM**
    - Create a table.
    - Calculation of energies (potential, elastic, kinetic, dissipated..).
    - Calculation of integrals or averages.
- Handling Fields and Tables: **CREA_CHAMP/ CREA_RESU/ CREA_TABLE/ CALC_TABLE**.
  - **CREA_CHAMP**: Create fields of quantitates champ_gd.
  - **CREA_RESU**: Create a DS resultat from fields.
  - **CREA_TABLE**: Create a table from a function or a list of real.
  - **CALC_TABLE**: Manipulate data’s from tables like a spreadsheet.
- Print a Result or Fields: **IMPR_RESU**.
  - Print:
    - Meshes.
    - Fields of quantities.
    - Data contained in a DS.
  - Different formats:
    - Listing: FORMAT = ‘**RESULTAT’** and ‘**ASTER’**
    - MED: FORMAT = ‘**MED’**
    - GMSH FORMAT = ‘**GMSH’**
    - CASTEM FORMAT = ‘**CASTEM’**
    - IDEAS FORMAT = ‘**IDEAS’**
- Print a Table.
  - Print the contents of a table in a listing or an Excel file.
  - Also plot curves.
  - Selection:
    - **NOM_PARA**: Choice of columns to print.
    - **FILTRE**: Choice of lines to print (some lines checking criteria).
    - **TRI**: Choice of the order of rows to print (ascending or descending).
    - **FORMAT**: Choice of print format.

Where to Find Information

- Access to all documentations of Code_Aster.
  - U documents: Usage.
    - Using the commands.
    - Methodological documents.
    - Documents for starting:
      - [U1.00.00] Getting started with Code_Aster
      - [U2.02.00] Introduction to Code_Aster
  - V documents: Validation.
    - One test-case = one V document.
    - Description of the problem and the reference solution.
  - R documents: Reference.
    - Theory related to methods and algorithms of the code.

- Tools for the community.
  - The forum is:
    - A place of exchange and mutual assistance for Code_Aster users.
    - Exchange of best practices, tips. Sharing of experience.

Useful Links

- www.code-aster.org → Official Code_Aster WebSite.
- www.salome-platform.org → Official Salome Platform WebSite.
- http://code-aster.org/forum2/ → Official Code_Aster, Salome Meca Forums.
- http://www.code-aster.org/spip.php?rubrique19 → Code_Aster, Salome_Meca Documentation.
- http://www.code-aster.org/spip.php?rubrique68 → Code_Aster, Salome_Meca Training.
- http://www.code-aster.org/spip.php?rubrique22 → Code_Aster, Salome_Meca Support.

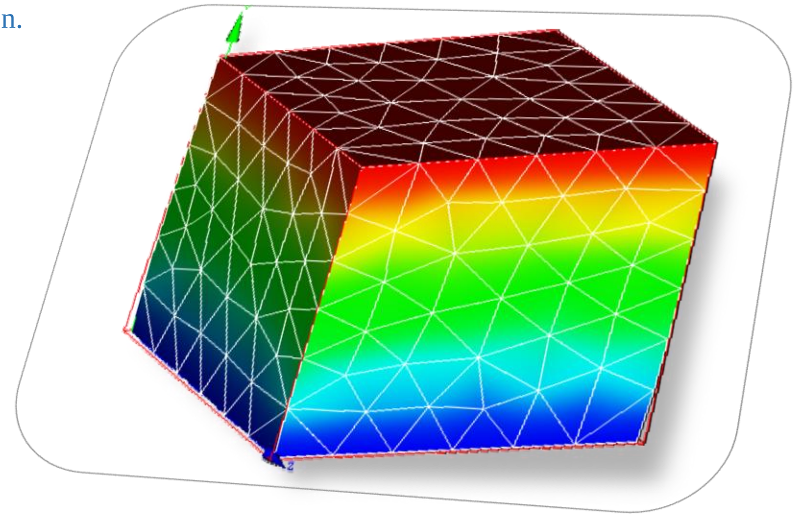
- http://caelinux.com/CMS/index.php?option=com_content&view=article&id=55:caelinux-2013&catid=69&Itemid=41 → Download CAELinux, Salome_Meca WebSite.
- https://www.virtualbox.org/ → VirtualBox Download WebSite.
- http://caelinux.org/wiki/index.php/Doc:CAETutorials → Code_Aster, Salome_Meca Tutorials.

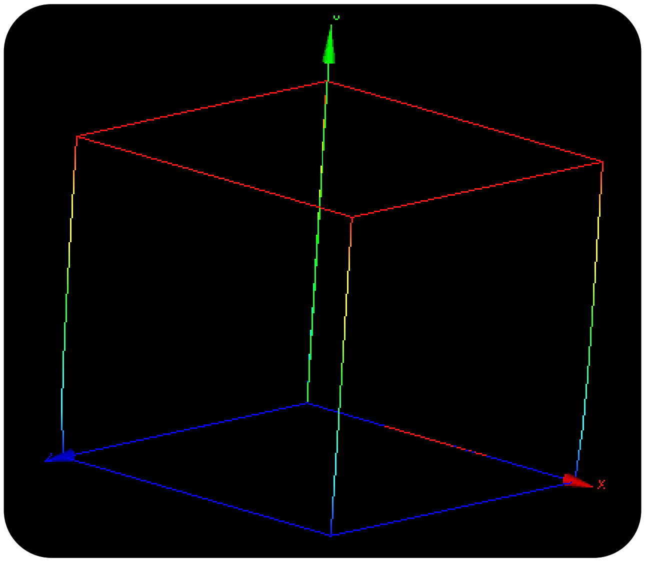
Personal Difficulties During Salome_Meca, Code_Aster Study

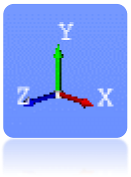
- The first problem I faced concerns Salome_Meca Post-Processing Module. I noticed that even though my mesh and my command file where absolutely correct, the platform was running the analysis BUT Post-Pro branch didn’t appear after the analysis. I figured later on, that this was a problem concerning the names I was giving to the folder, files, command files, aster study. That’s why you should avoid giving difficult names or even adding a space between letters, stay simple and also avoid symbols.
- Secondly, Geometry Viewer mode will in most cases get stuck. You will need to reset it everytime that happens, I found a way by closing the Geometry viewer, activating Mesh Module and then re-activate Geometry Module without making any changes.
- Lines beginning with “#” are comments. Also, many things are explained directly in the code with commented lines at the right place. A line with [U4.81.03] for example, means that the following command or operator is described in the U4.81.03 document in Code_Aster documentation available at the links given above.
- Another important note, is that within the command file we must not use concept names more than 8 characters.

Chapter_I: Cube

In the first chapter we will get familiar with the software concepts and in general with the idea of Finite Element Analysis Method. We will work on a Cube geometry to easy our prelusion.







Step 1: Purpose of the FE Analysis/ Description of the problem

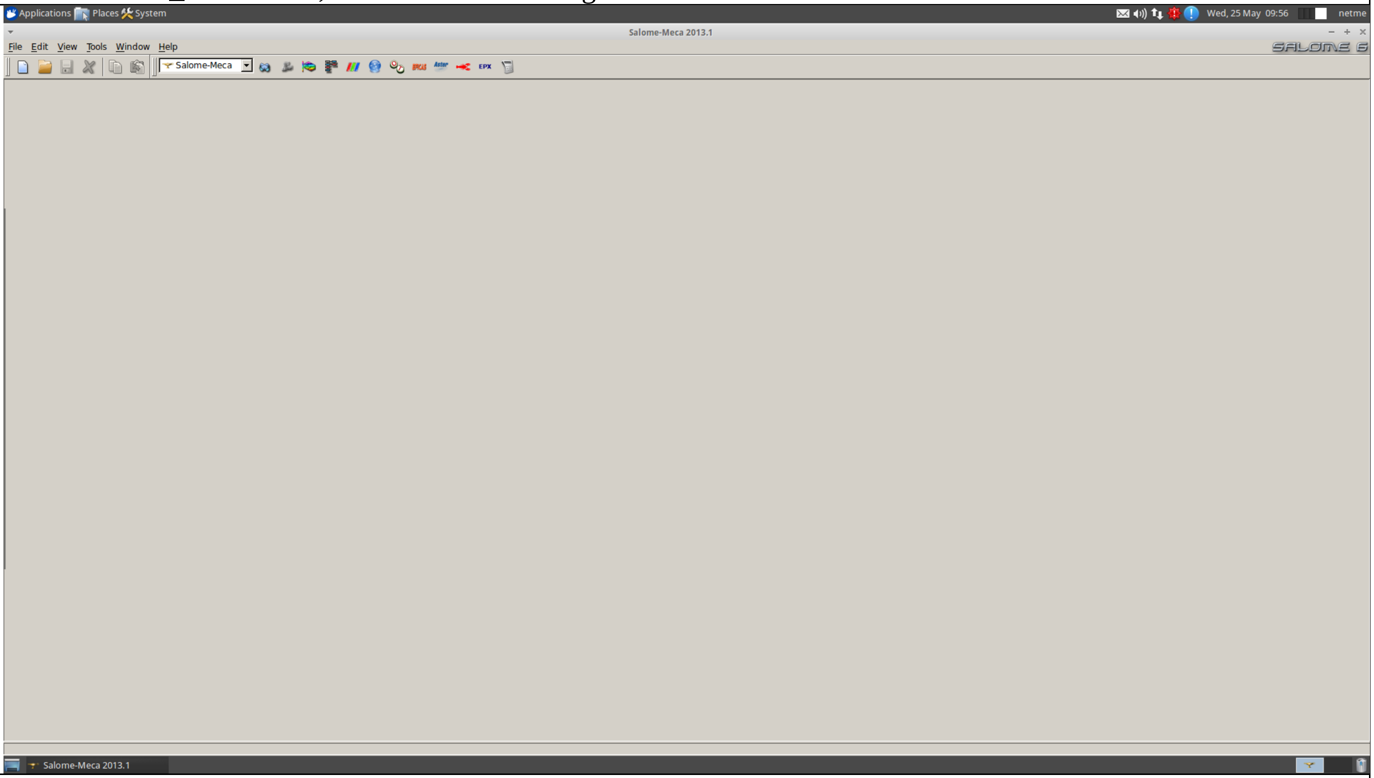
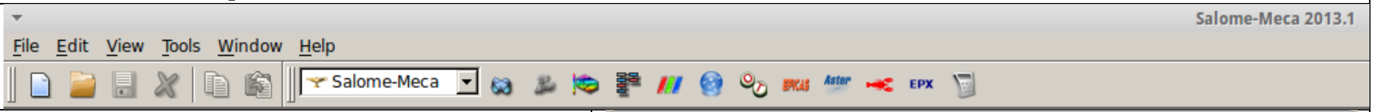
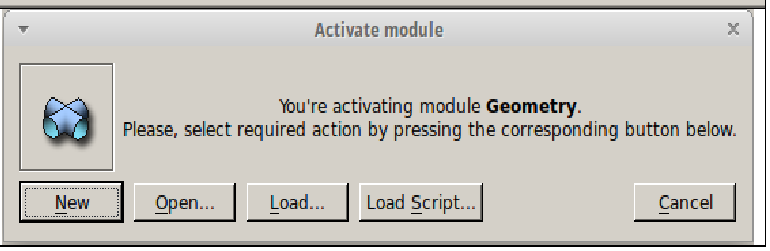
The purpose of this Finite Element Analysis is to make an outset of the Finite Element Analysis Method, and also getting to know the concepts of our Open_Source software, Salome_Meca.

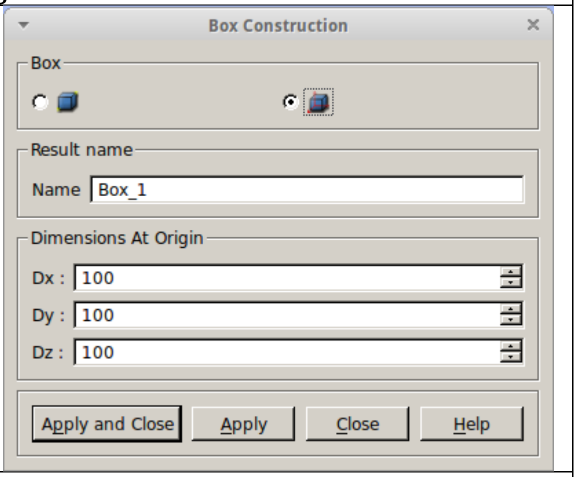
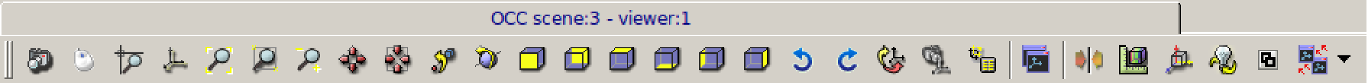
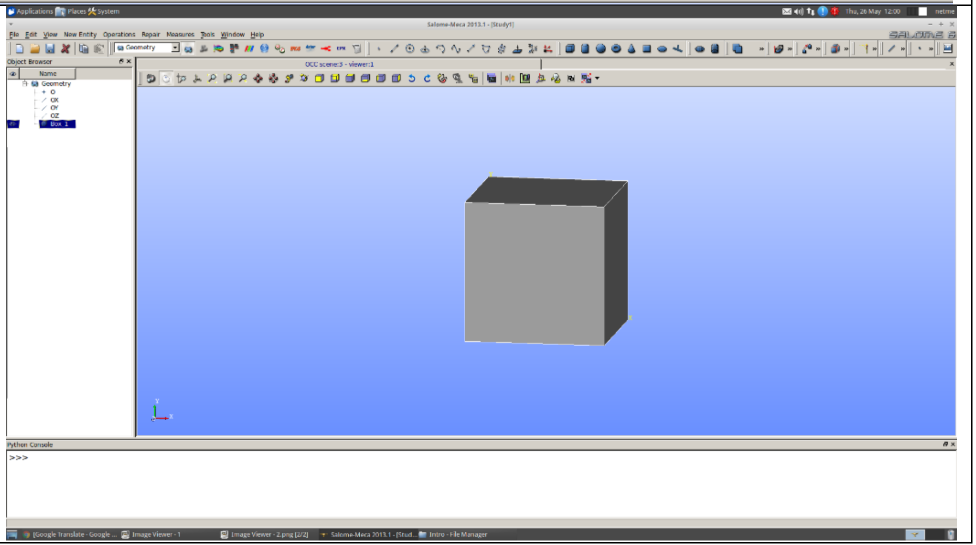
Step 2: Input Values for the FE Analysis

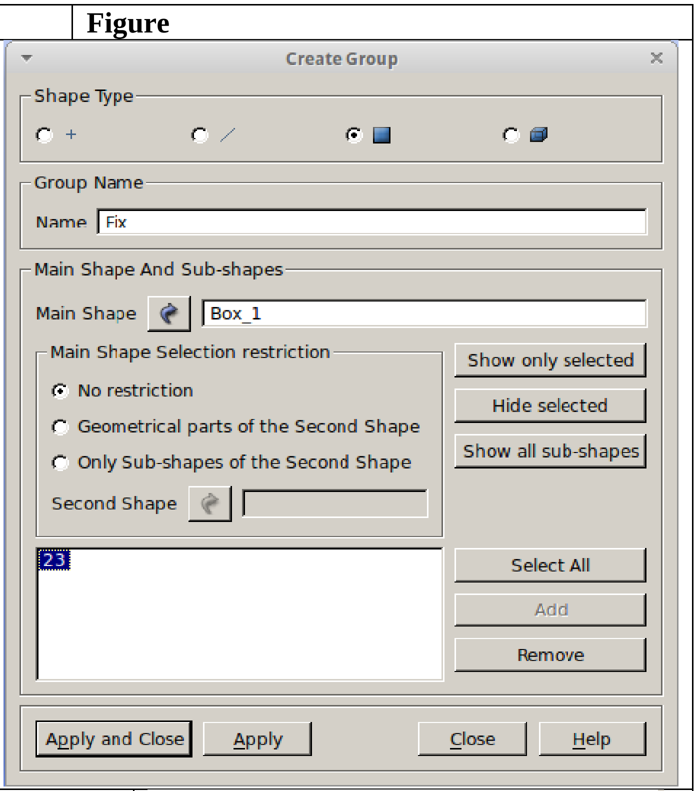
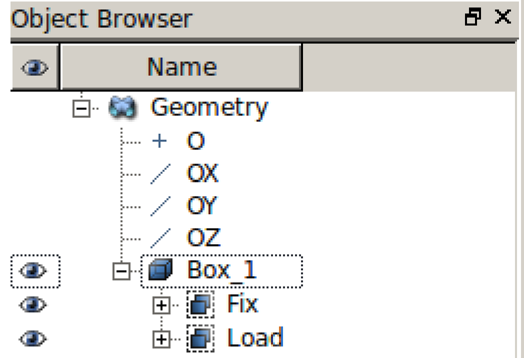
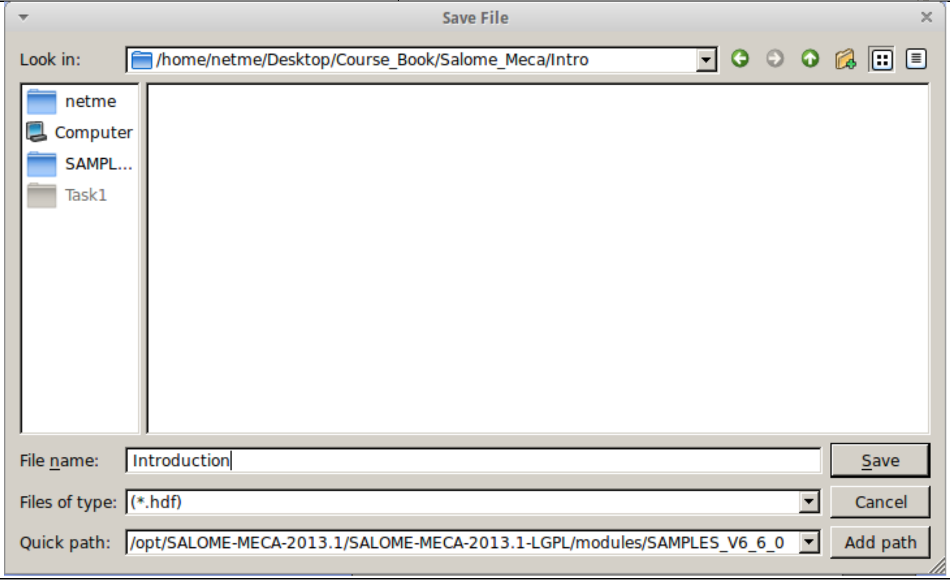
Cube Dimension:	100mm x 100mm x 100mm;
Young’s Modulus:	2.1e5 MPa;
Poisson’s Ratio:	0.3;
Pressure Applied on the “Top” face:	i) 5 MPa; ii) 50 MPa;

Two different load cases.

Step 3: Model Geometry

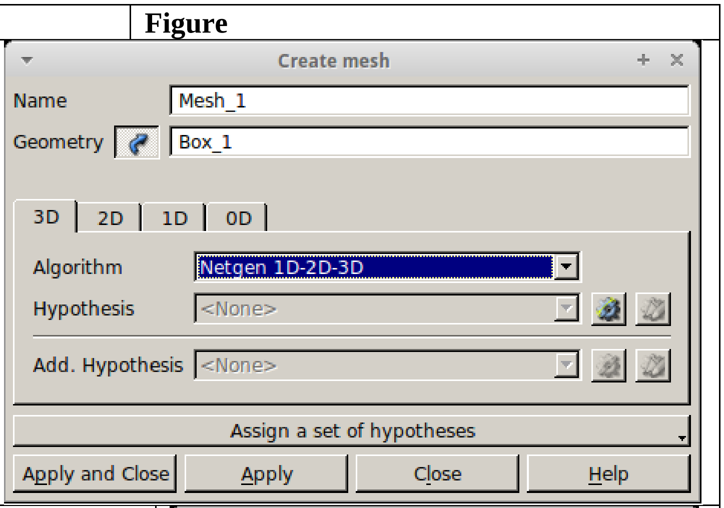
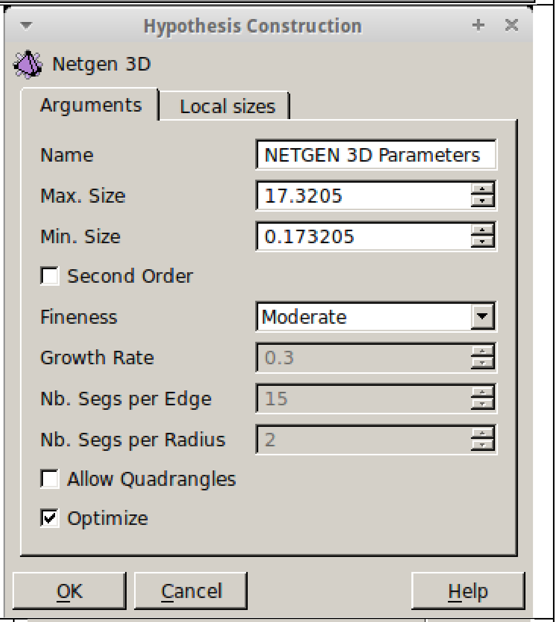
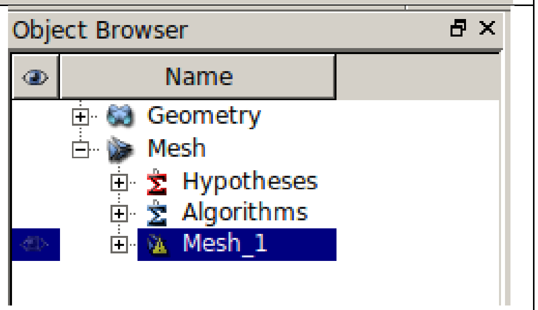
#	Description	Figure
1.	After installing Salome_Meca in your Ubuntu system we are ready to run the application. There are two different ways to start Salome_Meca. First one is to open up a terminal and go to the folder where Salome_Meca is installed and type *./runAppi* Or double-click on Salome_Meca icon on desktop. When Salome_Meca loads, it will look like the figure below.	
2.	 Applications Places System Wed, 25 May 09:56 netme Salome-Meca 2013.1 File Edit View Tools Window Help Salome-Meca SALOME 6 Salome-Meca 2013.1	
3.	The figure below shows the shape of Salome_Meca toolbar. The first line has a pretty common menu bar: File, Edit, View, Tools, Window, Help. The second line, except the *New*, *Open*, *Save*, *Close*, *Copy and Paste* tools, consists of (1) *Geometry*, (2) *Mesh*, (3) *Post-Pro*, (4) *YACS*, (5) *ParaViS*, (6) *Jobmanager*, (7) *Openturns*, (8) *Eficas*, (9) *Aster*, (10) *Homard*, (11) *Europleax*, (12) *MED*, and a drop-down list which contains all of the above.	
4.	 Salome-Meca 2013.1 File Edit View Tools Window Help Salome-Meca EFICAS Aster EPX	
5.	Let's start drawing our model. Select Geometry either from the drop-down list or by clicking on the icon. Select "*New*" from the pop-up window that appears. *See figure on the right*	 Activate module New Open... Load... Load Script... Cancel You're activating module **Geometry**. Please, select required action by pressing the corresponding button below.

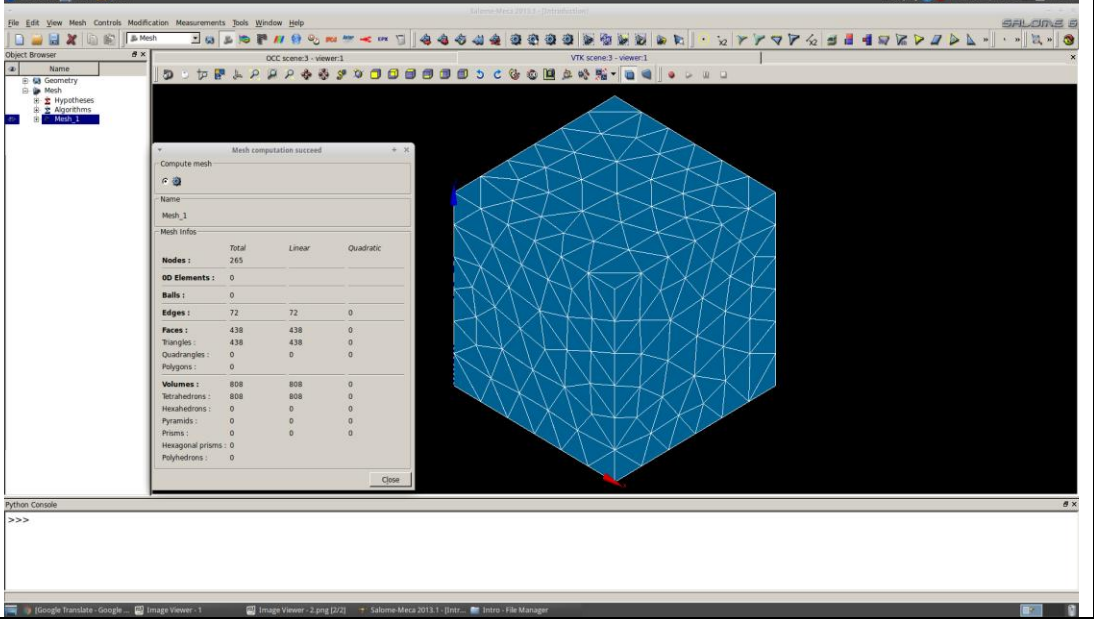
#	Description	Figure
6.	Click on New Entity → Primitives → Box from the toolbar. Fill in the gaps like the figure on the right. *Finish with "Apply and Close"*	 Box Construction Box ○ ● Result name Name: Box_1 Dimensions At Origin Dx : 100 Dy : 100 Dz : 100 Apply and Close · Apply · Close · Help
7.	Now in the viewer tab we got a new menu bar, if you move your mouse to each one of these icons it will show the name of each button. Most of those options are used to change your geometry's view, either by zooming in and out, rotating the model or even more.	
8.	 	
9.	Easy as it may seem, you have created your geometry and it is time to proceed to the analysis. First we need to create groups which will be a huge help to us as we continue through the FE Analysis.	

#	Description	Figure
10.	Select Box_1 from the Object Browser. Got to New Entity → Group → Create from the toolbar and fill the gaps as shown in the figure. Choose the 3rd option to select faces. Name the group with a proper name (we used Fix because we are targeting the fixed support face). Click on the bottom face to choose it and then press "*Add*". Once you select it, it will get highlighted with white edges. Finish with "*Apply*". Do the same procedure but now for the face that we want to apply the load (top face), you can name it Load.	 Create Group Shape Type: ○ + ○ ⁄ ● ■ ○ Group Name Name: Fix Main Shape And Sub-shapes Main Shape: Box_1 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape 23 Show only selected Hide selected Show all sub-shapes Select All Add Remove Apply and Close Apply Close Help
11.	After successfully completing all the previous steps, you should have the same Object Browser as shown on the figure on the right. The "*eye*" icons allow you to choose which part of our geometry will be shown on the Viewer tab. They are also used in every other Module (mesh, post-pro, ...)	 Object Browser Name Geometry O OX OY OZ Box_1 Fix Load
12.	Always save your work before moving to the next step/ module. Remember if you use meaningful names it is going to be really helpful later on. ***Note: Be extra careful when you name your work. Do not include space in your file names.***	 Save File Look in: /home/netme/Desktop/Course_Book/Salome_Meca/Intro netme Computer SAMPL... Task1 File name: Introduction Files of type: (*.hdf) Quick path: /opt/SALOME-MECA-2013.1/SALOME-MECA-2013.1-LGPL/modules/SAMPLES_V6_6_0

# Taking in consideration that you have completed all the previous steps correctly, we will continue to the Mesh part.

Step 4: Meshing Geometry

#	Description	Figure
1.	From the drop down list choose Mesh module. Select Box_1, go to the menu bar Mesh → Create Mesh. Keep the default name, Mesh_1. Make sure Box_1 is on the geometry option and choose Netgen 1D-2D-3D Algorithm. This algorithm option will automatically fill the 3D, 2D, 1D, 0D tabs.	 Create mesh Name Mesh_1 Geometry Box_1 3D 2D 1D 0D Algorithm Netgen 1D-2D-3D Hypothesis <None> Add. Hypothesis <None> Assign a set of hypotheses Apply and Close Apply Close Help
2.	We need to specify a hypothesis also. A hypothesis is a parameter that modulates the Algorithm. In this case we will adjust the Max. and Min. Size of the elements used by the NETGEN 1D-2D-3D Algorithm. Click on the gear and select Netgen 3D parameters. In the pop-up window leave everything as default (you will be explained in next chapters about those options). Click "*Ok*" to get back to the previous pop-up window. *Finish with "Apply and Close".*	 Hypothesis Construction Netgen 3D Arguments Local sizes Name NETGEN 3D Parameters Max. Size 17.3205 Min. Size 0.173205 ☐ Second Order Fineness Moderate Growth Rate 0.3 Nb. Segs per Edge 15 Nb. Segs per Radius 2 ☐ Allow Quadrangles ☑ Optimize
3.	Now your Object Browser should look like the figure on the right. If you noticed, Mesh_1 icon has an exclamation mark, which means that we may have set the mesh options but we still need to compute it. To do so, right-click on the Mesh_1 and select Compute.	 Object Browser Name Geometry Mesh Hypotheses Algorithms Mesh_1

#	Description	Figure
4.	If everything is well placed, after some seconds you will get a pop-up window which contains all of the Mesh information. Like number of Nodes, Edges, Faces, etc. In the figure on the right you can also see how our model will look after we compute the mesh.	

Mesh computation succeed

Compute mesh

Name

Mesh_1

Mesh Infos

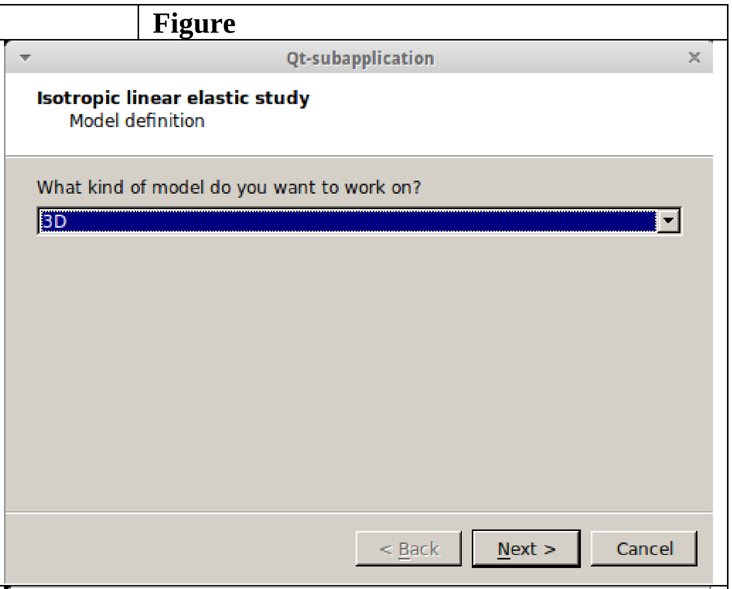
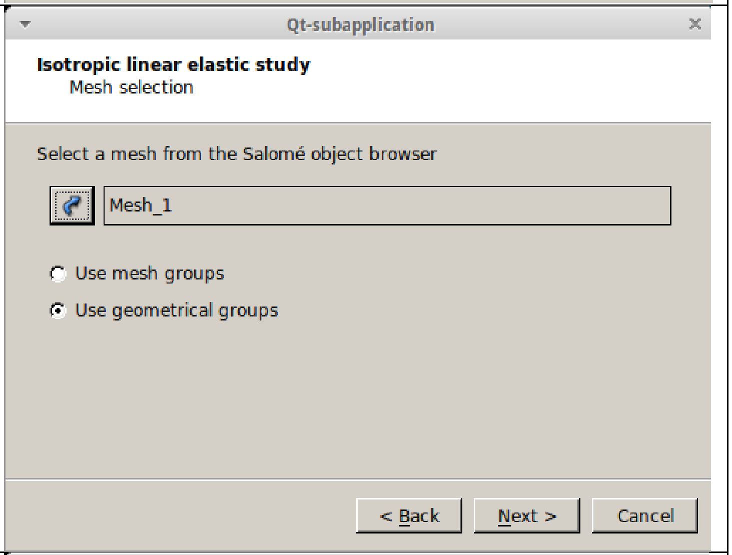
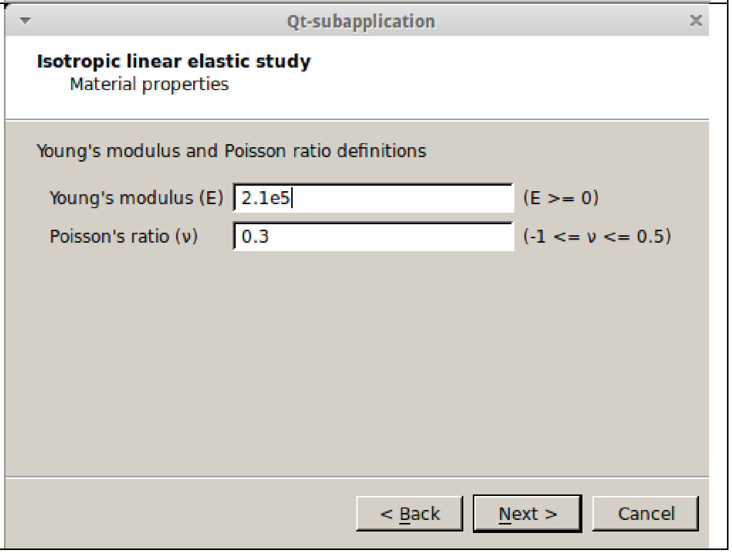
	Total	Linear	Quadratic
Nodes :	265		
0D Elements :	0		
Balls :	0		
Edges :	72	72	0
Faces :	438	438	0
Triangles :	438	438	0
Quadrangles :	0	0	0
Polygons :	0		
Volumes :	808	808	0
Tetrahedrons :	808	808	0
Hexahedrons :	0	0	0
Pyramids :	0	0	0
Prisms :	0	0	0
Hexagonal prisms :	0		
Polyhedrons :	0		

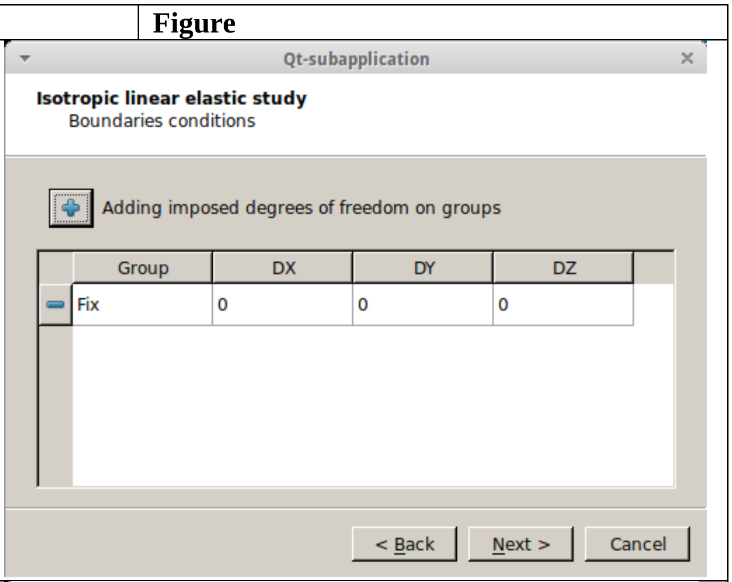
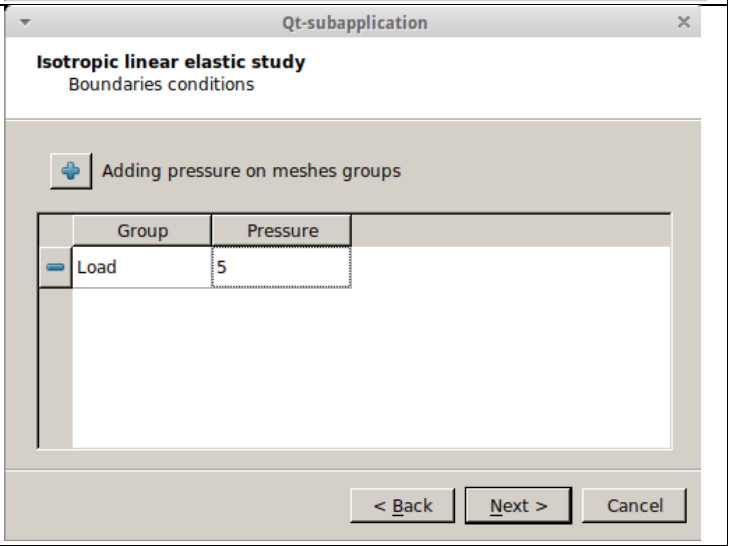
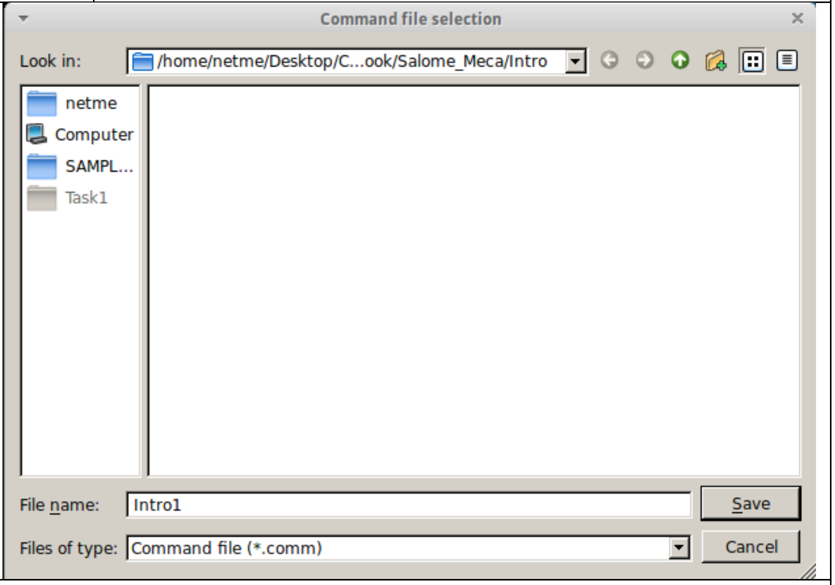
```
>>>
```

# That was the end of our meshing step. Remember to always save your work before going to the next step. In the next steps we will see how to build information for a Linear Static FE Analysis with the help of Salome_Meca Wizard.

Step 5,6,7,8 and 9: Salome_Meca Linear Static Analysis Wizard

# After completing our geometry, arranging the correct groups, using the most suitable algorithm and hypothesis for the mesh, now is time to make some more modification for out linear static analysis.

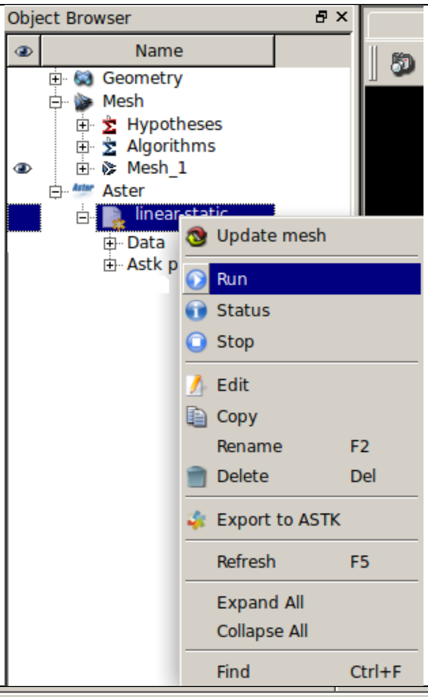
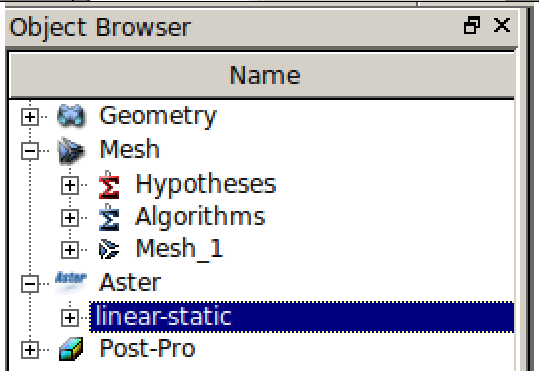
#	Description	Figure
1.	Select Aster Module from the drop down list or by clicking on the icon from the toolbar. Go to Aster → Wizards → Linear Elastic. Step 5: Define the type of model, select 3D like in the figure on the right. *Proceed with "Next"*	 Qt-subapplication **Isotropic linear elastic study** Model definition What kind of model do you want to work on? 3D
2.	Step 6: Select the mesh that will be used for this specific analysis. If for some reason Mesh_1 is not automatically chosen, press the blue arrow button and then select Mesh_1 from the Object Browser. "*Use geometrical groups*", the ones that we defined in Geometry Module. *Proceed with "Next"*	 Qt-subapplication **Isotropic linear elastic study** Mesh selection Select a mesh from the Salomé object browser Mesh_1 ○ Use mesh groups ● Use geometrical groups
3.	Step 7: Define the material properties that we will use in our analysis. Add Young's modulus and Poisson's ratio as shown in the figure on the right. Our dimensions are in millimeters so our Young's modulus should be in MPa. *Proceed with "Next"*	 Qt-subapplication **Isotropic linear elastic study** Material properties Young's modulus and Poisson ratio definitions Young's modulus (E) 2.1e5 (E >= 0) Poisson's ratio (ν) 0.3 (-1 <= ν <= 0.5)

#	Description	Figure
4.	Step 8: Input Geometric Boundary Conditions. This is where the geometry groups we created before will come in handy. We want fixed support in the *Fix* group. Fixed support has zero degrees of freedom for the selected group (in case of volumetric geometry- this leads to discretization by solid elements), so fill in the gaps according to the figure on the right. *Proceed with "Next"*	 **Isotropic linear elastic study** Boundaries conditions Adding imposed degrees of freedom on groups Group DX DY DZ Fix 0 0 0
5.	Step 9: Input Load Boundary Condition. Select Load and input 5 MPa if it is not already selected. Load is the top face group we created in the Geometry Module. *Proceed with "Next"*	 **Isotropic linear elastic study** Boundaries conditions Adding pressure on meshes groups Group Pressure Load 5
6.	We have defined all the information needed for the FE Analysis. We also have created our command file (.comm), that passes on the information we just input in the wizard, to Code_Aster. Click on the icon to save the (.comm) file in the same folder that the .hdf file is and, for your comfort, name it with a meaningful name. *Click "Save"* *End with "Finish"*	 Look in: /home/netme/Desktop/C...ook/Salome_Meca/Intro File name: Intro1 Files of type: Command file (*.comm)
7.	A new branch has been created to the Object Browser. Open "*Aster*" and you will see the "*linear-static*" underneath.	

# Summarizing our work so far, we have created our geometry, separated the boundary groups, computed the mesh and we created the command file containing all the necessary information about our analysis. Only thing left to do is ….

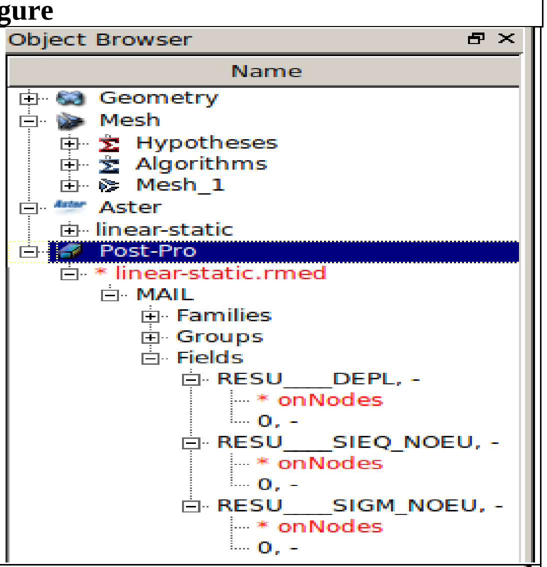
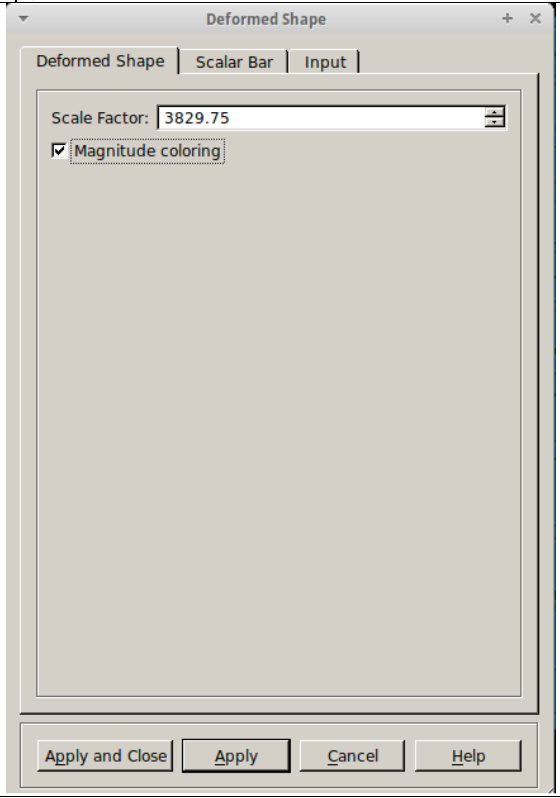
Step 10: Running the Analysis

# Salome_Meca makes it easy for beginners to create and run an analysis (always depending on the kind of the analysis). The end results (output) can be changed based on the results we are after, by editing the command file. But for this introductory analysis we will use defaults.

#	Description	Figure
1.	Right-click on "*linear-static*" in the Object Browser and select "*Run*" A new window will pop-up displaying lots of different things. The graphic environment is the software's way to show procedures, steps of solution by text output. The software now calculates all the information we entered before.	 Object Browser Name Geometry Mesh Hypotheses Algorithms Mesh_1 Aster linear-static Data Astk p Update mesh Run Status Stop Edit Copy Rename F2 Delete Del Export to ASTK Refresh F5 Expand All Collapse All Find Ctrl+F
2.	If everything goes well, a new branch named "Post-Pro" will appear in the Object Browser. If Post-Pro branch doesn't appear, it means that something went wrong with the analysis. When that happens, the information about the error is written in text file with suffix .mess in the directory, where the task was saved. If you are not sure about the mistake, it would be better to re-do all the steps from scratch. There is another way of correcting those errors but we will see it in the next chapters.	 Object Browser Name Geometry Mesh Hypotheses Algorithms Mesh_1 Aster linear-static Post-Pro

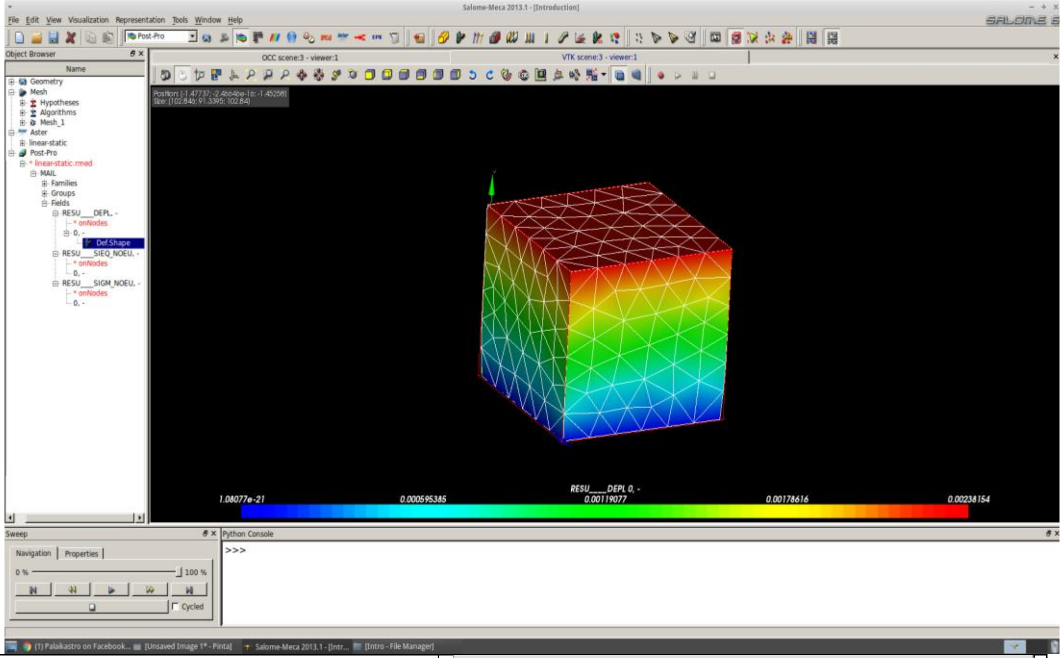
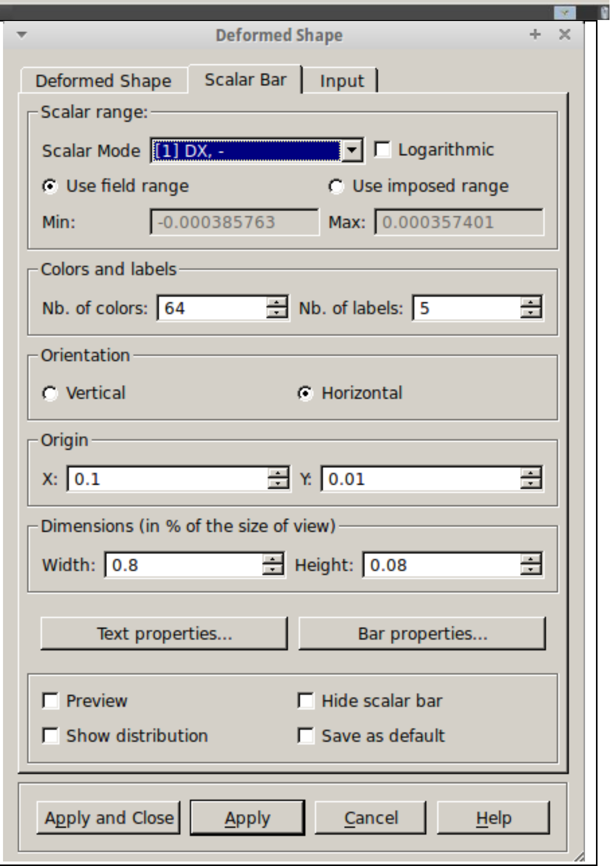
# Post-Pro branch is now on the Object Browser which means that the results are ready to be evaluated. Depending on the type of the analysis the calculation could take from a few minutes to several hours. Next step is to visualize the results.

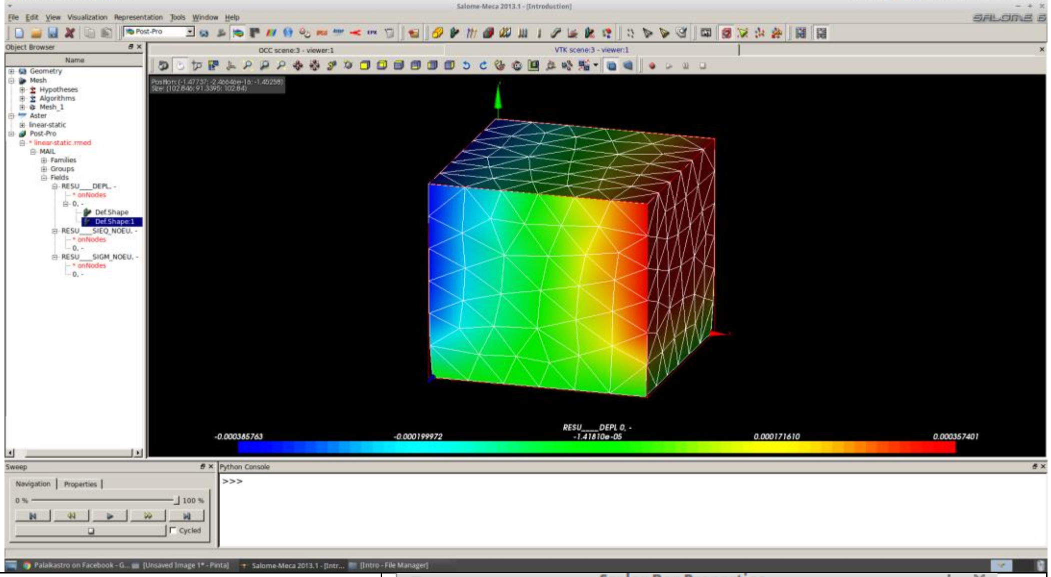
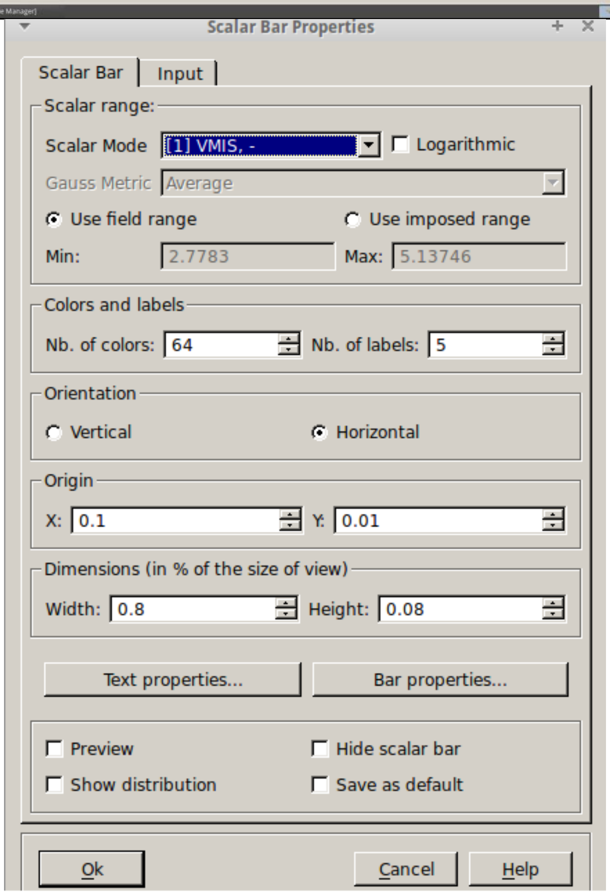
Step 11: Post Processing the results

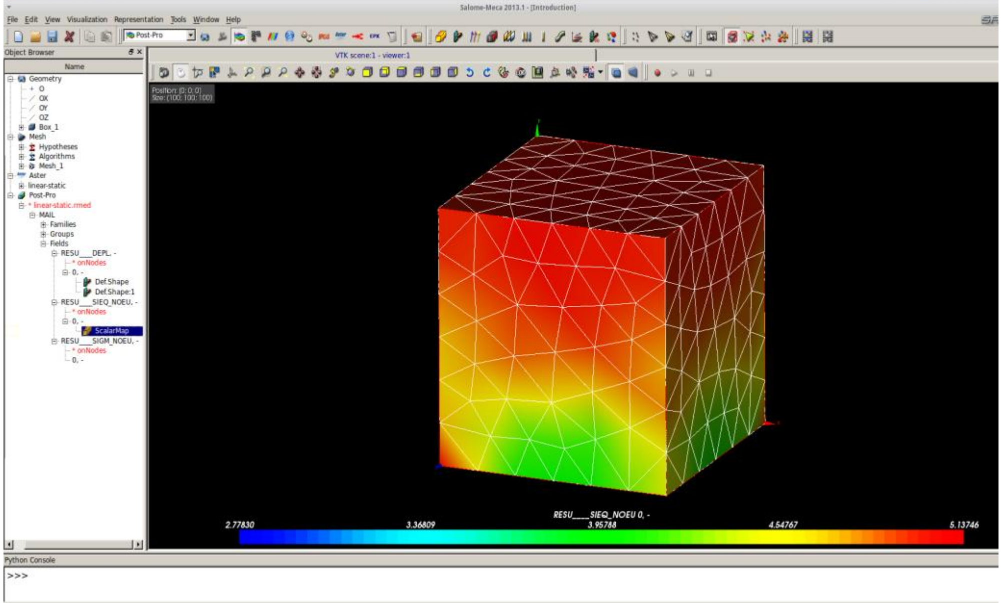
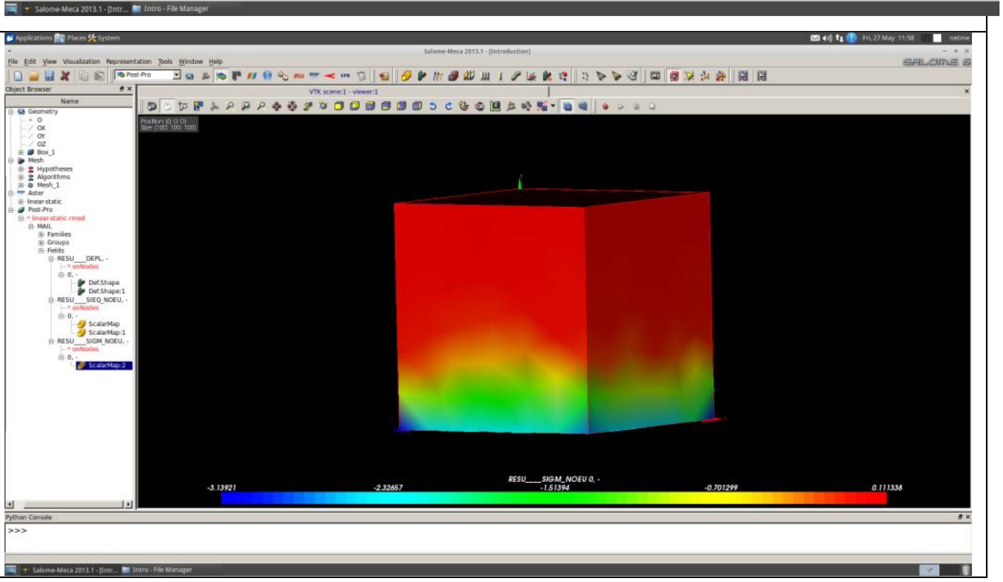
#	Description	Figure
1.	Select Post-Pro from the drop down list to enter Post-Pro Module. In the Object Browser we can now see three branches in the parent branch of field. These branches contain the results of the analysis, and the only thing left to do is explain to the software the way to visualize those results.	
2.	RESU___DEPL → Result of the Displacement of the Nodes. Open this branch, right click on "0,-" and select Deformed Shape. Check the Magnitude coloring in the pop-up window and keep the default scale factor. You can go through the other 2 tabs Scalar Bar, Input, but right now we won't need to change something else in those tabs. *Finish with "Apply and Close"*	

Scale Factor: 3829.75

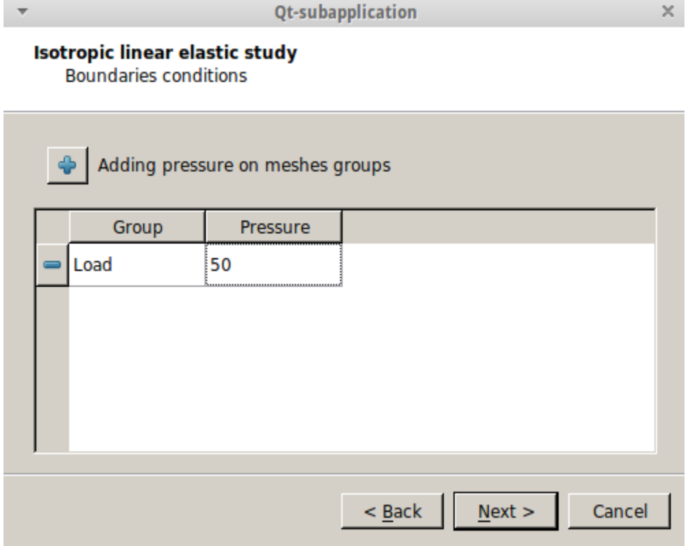
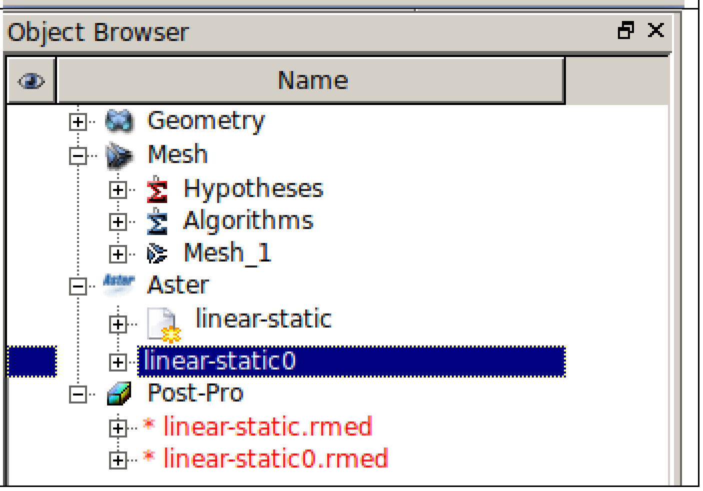
☑ Magnitude coloring

#	Description	Figure
3.	A new branch appears inside RESU___DEPL. With right-clicking on the Def.Shape you get plenty of options. First of all select the "*Show Only*" option and then right-click → Representation → SurfaceFrame. As you can see from the figure at the bottom of the window, a graph of Deflection with minimum and maximum values is shown.	 Salome-Meca 2013.1 - [Introduction] Object Browser Geometry Mesh Hypotheses Algorithms Mesh_1 Aster linear-static Post-Pro linear-static.rmed MAIL Families Groups Fields RESU___DEPL, - onNodes 0, - Def.Shape RESU___SIEQ_NOEU, - RESU___SIGM_NOEU, - OCC scene:3 - viewer:1 VTK scene:3 - viewer:1 RESU___DEPL 0, - 1.08077e-21 0.000595385 0.00119077 0.00178616 0.00238154 Sweep Navigation Properties Python Console >>>
4.	Now in this step, I will show you how to add more than one different types of visualization in the study if required. Right-click again on "0,-" select Deformed Shape, check the Magnitude Box with default Scale Factor. Click on Scalar Bar tab, go to Scalar Mode and select DX from the given option, as shown in the figure. *Finish with "Apply and Close"*	Deformed Shape Scalar range: Scalar Mode [1] DX, - ☐ Logarithmic ● Use field range ○ Use imposed range Min: -0.000385763 Max: 0.000357401 Colors and labels Nb. of colors: 64 Nb. of labels: 5 Orientation ○ Vertical ● Horizontal Origin X: 0.1 Y: 0.01 Dimensions (in % of the size of view) Width: 0.8 Height: 0.08 ☐ Preview ☐ Hide scalar bar ☐ Show distribution ☐ Save as default Deformed Shape Scalar Bar Input Text properties... Bar properties... Apply and Close Apply Cancel Help

#	Description	Figure
5.	Follow the same steps as before to make it visible and with the use of SurfaceFrame. Here you are able to see how the cube deforms in X direction. You can also see in Y,Z direction by changing the DX option from the drop down list given in the Scalar Bar tab.	 -0.000385763 -0.000199972 RESU___DEPL 0, - -1.41810e-05 0.000171610 0.000357401
6.	We can see the stress results now, and the points where those stresses reach maximum and minimum values. RESU___SIEQ_NOEU is the results (RESU) of the stress (SI..) Equivalent (..EQ) or equivalent stress at nodes (NOEU). Again right-click on “0,-“ choose Scalar Map and from the pop-up window select VMIS (Von Mises), rest keep default. *Finish with “Ok”*	 Scalar Bar Properties Scalar Bar | Input Scalar range: Scalar Mode: [1] VMIS, - ☐ Logarithmic Gauss Metric: Average ● Use field range ○ Use imposed range Min: 2.7783 Max: 5.13746 Colors and labels Nb. of colors: 64 Nb. of labels: 5 Orientation ○ Vertical ● Horizontal Origin X: 0.1 Y: 0.01 Dimensions (in % of the size of view) Width: 0.8 Height: 0.08 Text properties... Bar properties... ☐ Preview ☐ Hide scalar bar ☐ Show distribution ☐ Save as default Ok Cancel Help

#	Description	Figure
7.	A new branch has appeared now at the Object Browser, right-click → Show Only, right-click → Representation → SurfaceFrame to get an understandable view. Notice that at the bottom of the window a graph with minimum and maximum Von Mises values is shown. As in the deflection branch we can also change the fashion of showing (to other similar stresses) like TRESCA or VMIS_SG. All you have to do is follow the same steps as before and from the pp-up window instead of choosing VMIS, select any other of the given options.	 Salome-Meca 2013.1 - [Introduction] Object Browser VTK scene:1 - viewer:1 RESU___SIEQ_NOEU 0, - 2.77830 3.36809 3.95788 4.54767 5.13746 Python Console >>>
8.	Following the same steps as before for the RESU___SIGM_NOEU branch, RESU___SIGM_NOEU is the result of the stress or SIGMa at Nodes. Here we have 6 different stresses, Sx, Sy, Sz, Sxy, Sxz, Syz.	 Salome-Meca 2013.1 - [Introduction] Object Browser VTK scene:1 - viewer:1 RESU___SIGM_NOEU 0, - -3.13921 -2.32657 -1.51394 -0.701299 0.111338 Python Console >>>

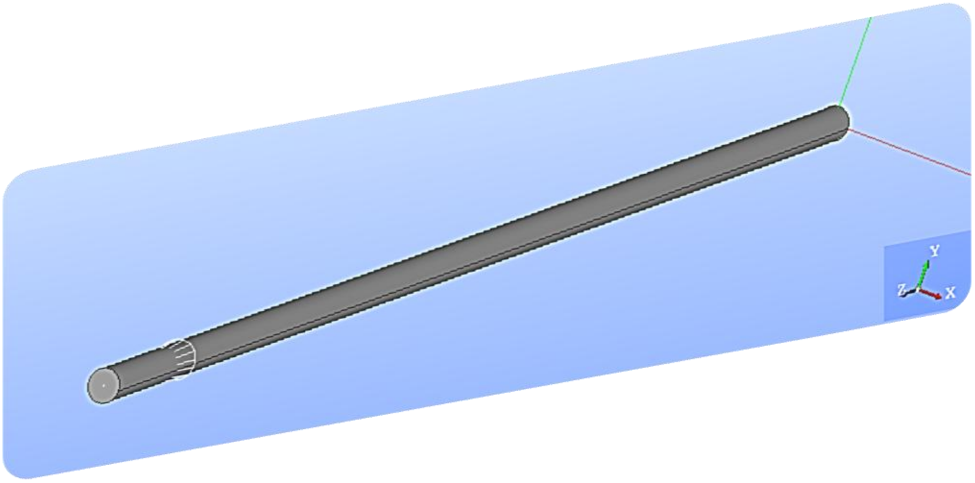
# We are finished with our first load case, now let’s see what changes we have to make to solve our second case scenario.

#	Description	Figure
1.	We won't have to start from scratch since we are going to use the same geometry and mesh. Choose Aster Module from the drop down window. Make sure Mesh_1 is selected and follow the exact same steps we did for the first load case. **BUT**, as it is shown in the figure on the right, when it comes to input the value of the load you will have to input 50 MPa. *Continue with saving with different name and finish.*	 Qt-subapplication Isotropic linear elastic study Boundaries conditions Adding pressure on meshes groups Group Pressure < Back Next > CancelLoad 50
2.	As we can see from the figure, a new analysis is added to Aster branch. Right-click → Run and if everything is correct there will also be a new branch in Post-Pro branch. Repeat the same steps as in the previous analysis to visualize the results in this load case.	 Object Browser Name Geometry Mesh Hypotheses Algorithms Mesh_1 Aster linear-static linear-static0 Post-Pro linear-static.rmed linear-static0.rmed

# This task was a complete FE Analysis which is very helpful to understand the steps we follow in Finite Element Analysis.

Chapter_II: Cylinder Rod

In this next chapter we are going to get familiar with Code_Aster and how programming affects Salome_Meca software.
We will study a Cylinder Rod case and use command line to set the necessary parameters.



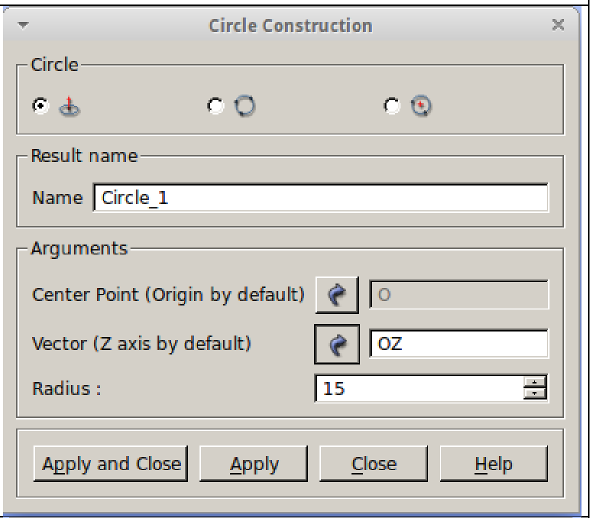
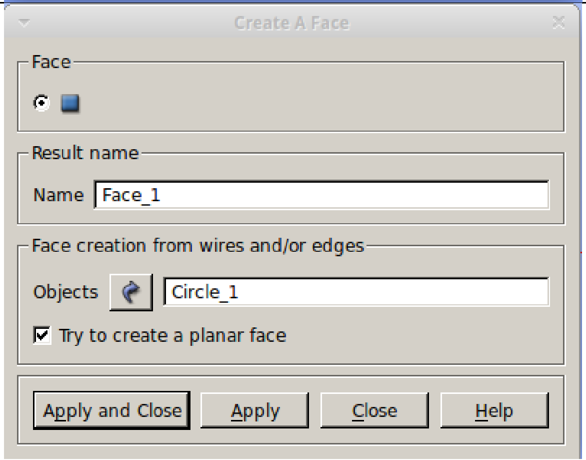
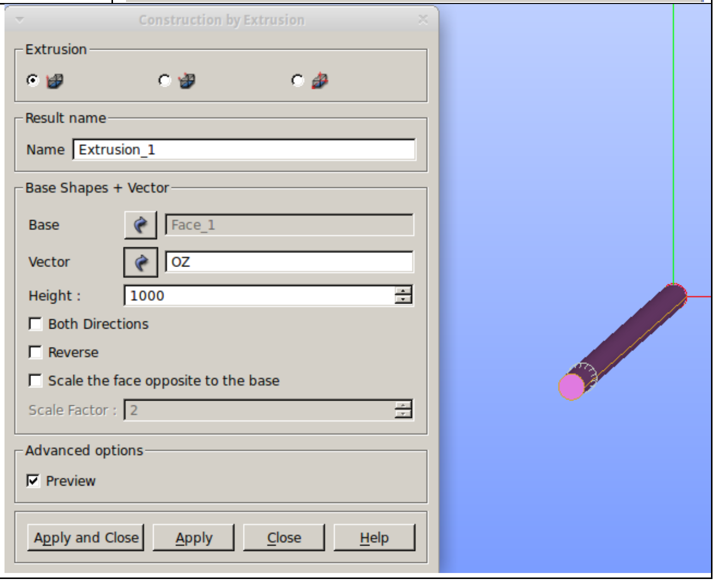
Step 1: Purpose of the FE Analysis/ Description of the problem

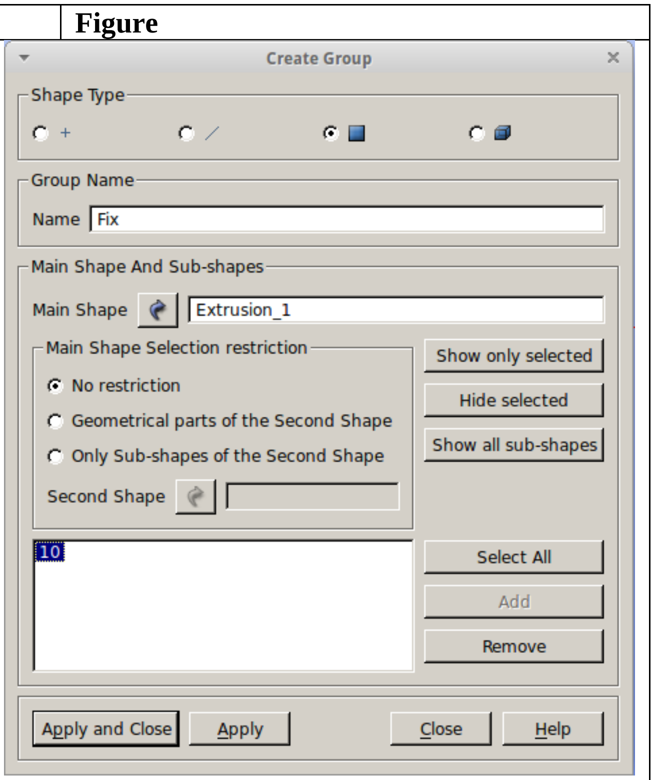
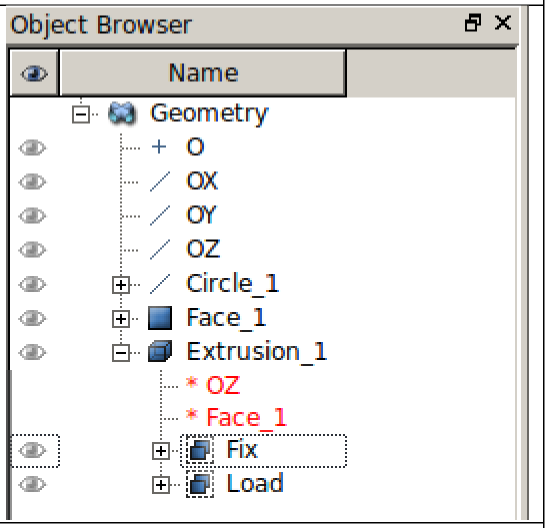
The purpose of this Finite Element Analysis is to get one step close to understanding Finite Element Analysis Method, and also getting to know the concepts of our Open_Source software, Salome_Meca and Code_Aster.

Step 2: Input Values for the FE Analysis

Cylinder Dimensions:	l= 1000mm, r= 15mm;
Young’s Modulus:	2e5 MPa;
Poisson’s Ratio:	0.3;
Pressure Applied on the “right edge” face:	500 N;

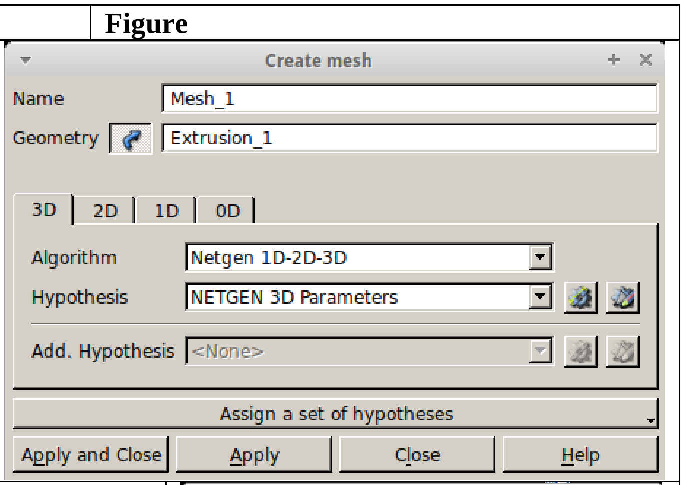
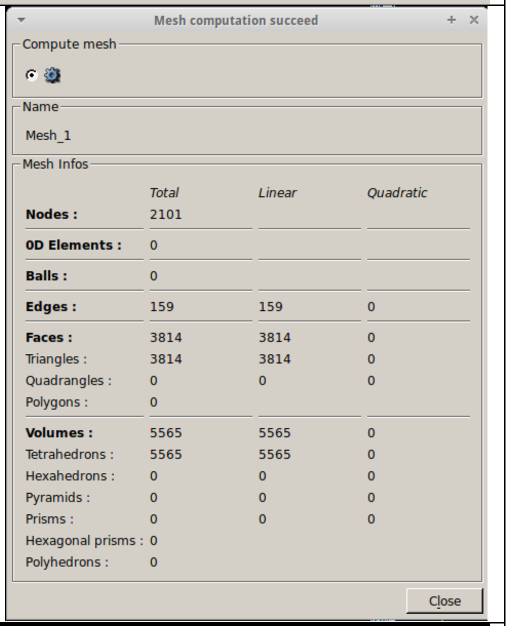
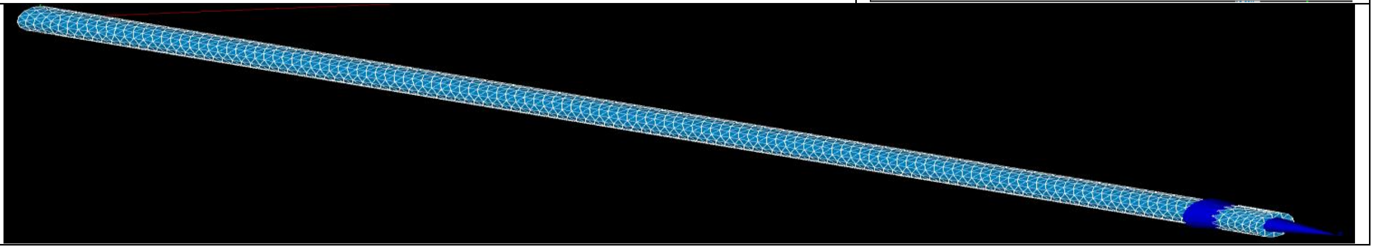
Step 3: Model Geometry

#	Description	Figure
1.	Follow the same procedure you did in the previous chapter. Starting from Step 3: Model Geometry #1 to #6.	
2.	Click on New Entity → Basic → Circle in Salome_Meca toolbar. Fill in the gaps according to the figure on the right to create our Cylinder. *Finish with "Apply and Close"*	 Circle Construction Circle ● ○ ○ Result name Name Circle_1 Arguments Center Point (Origin by default) O Vector (Z axis by default) OZ Radius : 15
3.	Click again on New Entity → Build → Face and fill the blanks according to the figure on the right. *Finish with "Apply and Close"* *We made a face from our geometry, to be able to use the Extrusion command.	 Create A Face Face ● Result name Name Face_1 Face creation from wires and/or edges Objects Circle_1 ☑ Try to create a planar face
4.	Next step is creating our Solid Geometry model. Click again on New Entity → Generation → Extrusion. In the construction by extrusion window, fill in the gaps according to the figure on the right. Your geometry also should have a look like in the figure. *Finish with "Apply and Close"*	 Construction by Extrusion Extrusion ● ○ ○ Result name Name Extrusion_1 Base Shapes + Vector Base Face_1 Vector OZ Height : 1000 ☐ Both Directions ☐ Reverse ☐ Scale the face opposite to the base Scale Factor : 2 Advanced options ☑ Preview

#	Description	Figure
5.	We now have created our geometry and the following step would be defining our groups that will help us make the FE Analysis. Make sure Extrusion_1 is selected in the Object Browser. Click on New Entity → Group → Create In the Create Group window that pops-up select the third option for shape type and select the faces from the geometry. This first group will be used for support, so it will help if you name your groups, and general all your options in the Object Browser, with easy to remember names. I used “*Fix*” as my fixed support group. Fill in the gaps according to the figure on the right. Rotate the model so you can find the face where we need our fixed support. When you click on the face we are interested in, it will get highlighted with white edges. Clicking on the “Add” button, the face number will be added as shown on the figure. Repeat the same procedure for selecting the opposite face and name it “*Load*”. *Finish with “Apply and Close”*	 Create Group Shape Type ○ + ○ ● ○ Group Name Main Shape And Sub-shapes Main Shape Selection restriction Show only selected Hide selected Show all sub-shapes Select All Add Remove Apply and Close Apply Close Help Name: Fix Main Shape: Extrusion_1 ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape: 10
6.	Now your Object Browser should look like in the figure on the right. The eyes on each option hide/show those options in the viewer window.	 Geometry O OX OY OZ Circle_1 Face_1 Extrusion_1 OZ Face_1 Fix Load
7.	Now that we are finished, save your work to a destination of your choise and name it accordingly.	

Step 4: Meshing Geometry

# Step by step example of how to mesh our geometry is shown below, taking in consideration that you have completed our geometry and have saved it to a logical path with a meaningful name.

#	Description	Figure
1.	Pull down the category box or click on the mesh icon to enter to Mesh Module. Select Extrusion_1 from the Object Browser and click Mesh → Create Mesh and fill the pop-up window likewise. In the Hypothesis tab, you won't have to change anything, just keep the data default. *Finish with "Apply and Close"*	 Create mesh Name: Mesh_1 Geometry: Extrusion_1 3D | 2D | 1D | 0D Algorithm: Netgen 1D-2D-3D Hypothesis: NETGEN 3D Parameters Add. Hypothesis: <None> Assign a set of hypotheses Apply and Close Apply Close Help
2.	Right-click on Mesh_1 → Compute. If everything goes well, you will have the same pop-up window which includes the mesh information. *Click on "Close"* Your mesh should look like the figure below.	 Mesh computation succeed Compute mesh Name: Mesh_1 Mesh Infos: Total / Linear / Quadratic Nodes : 2101 0D Elements : 0 Balls : 0 Edges : 159 / 159 / 0 Faces : 3814 / 3814 / 0 Triangles : 3814 / 3814 / 0 Quadrangles : 0 / 0 / 0 Polygons : 0 Volumes : 5565 / 5565 / 0 Tetrahedrons : 5565 / 5565 / 0 Hexahedrons : 0 / 0 / 0 Pyramids : 0 / 0 / 0 Prisms : 0 / 0 / 0 Hexagonal prisms : 0 Polyhedrons : 0
3.		

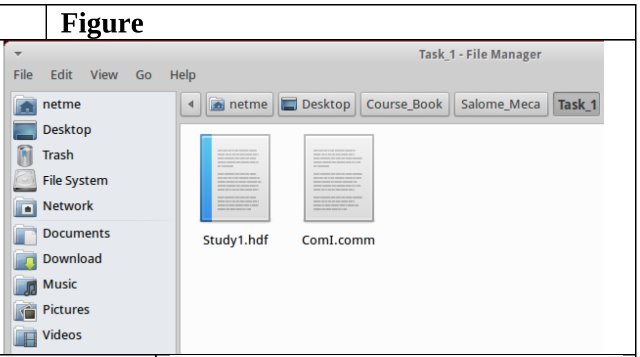
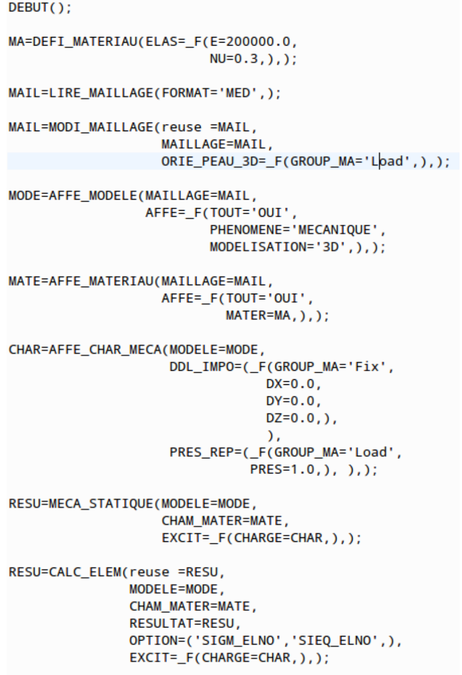
Step 5,6,7,8 and 9: Salome_Meca Linear Static Analysis Wizard

# In the previous chapter, in these steps we chose to use Salome_Meca Linear Static Wizard to create the command (.comm) file which would hold information about the analysis we wanted to do. BUT Linear Static Wizard doesn't give us the option to set our force in an horizontal way. In this case scenario we will use the Linear Static Wizard to create our command file and then we will modify it.

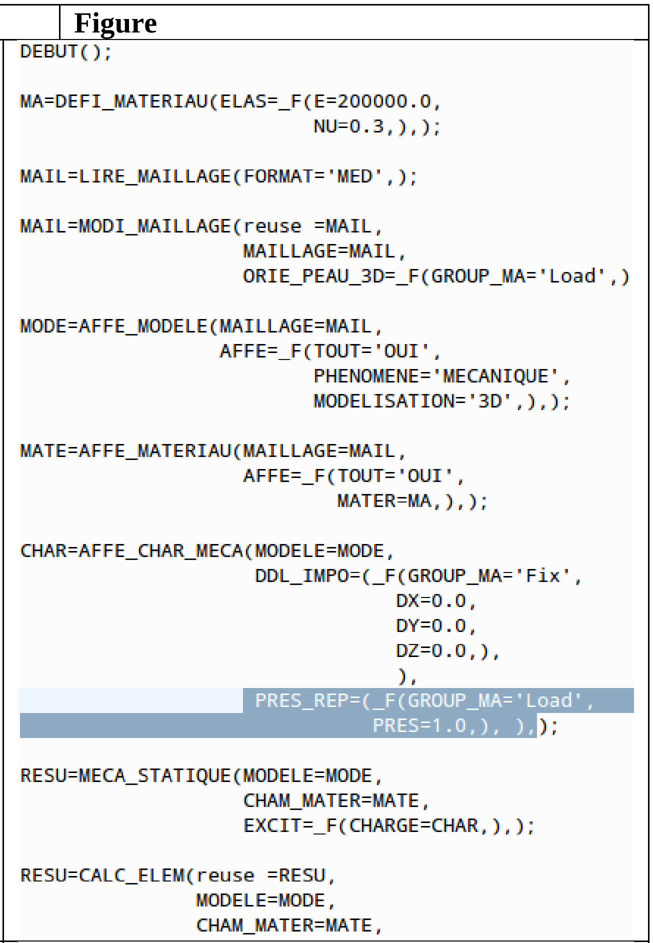
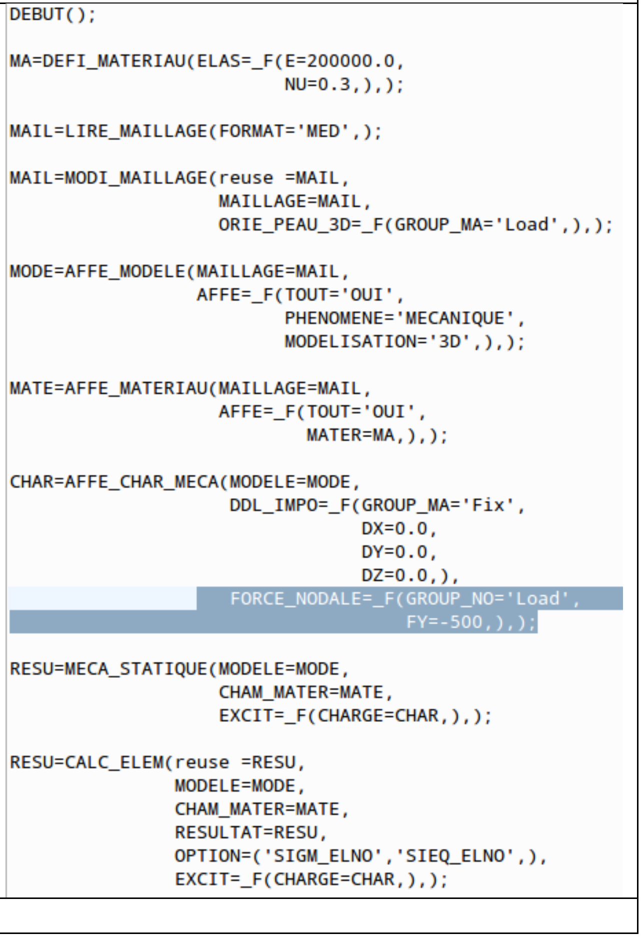
#	Description	Figure
1.	Click on Aster Module and make sure Mesh_1 is selected in the Object Browser. In Salome_Meca main toolbar, click Aster → Wizards → Linear Static and then a window will pop-up. Select "*3D*" as a type of Analysis, continue with "Next". Select Mesh_1 and choose "*Use of geometrical groups*", continue with "*Next*". Enter 2e05 (MPa) for Young's Modulus and 0.3 for Poison's Ratio, continue with "*Next*". For Geometric Boundary Conditions select "*Fix*" and give 0,0,0 values, continue with "*Next*". For Load Boundary Conditions, it doesn't matter what value you input here because we are going to modify it in the next steps, continue with "*Next*". Save the file by giving it an appropriate name. "*Finish*".	

# Now we have completed the steps to generate a study and create the command (.comm) file as we did in the previous chapter. Our next step is to edit the command file and adjust our force correctly.

Step 9: Manual editing of command (.comm) file

#	Description	Figure
1.	Search and open the folder where you saved the command file, it should be in the same folder where your .hdf file is located. If your Ubuntu installation is unchanged, this file loads with "*medit*" text editor, by default.	 Task_1 - File Manager File Edit View Go Help netme Desktop Trash File System Network Documents Download Music Pictures Videos netme Desktop Course_Book Salome_Meca Task_1 Study1.hdf ComI.comm
2.	When the text editor opens, it should look like the figure on the right. You don't have to worry or get scared what all those lines after lines mean or how they interact with Salome_Meca or Code_Aster, yet. The modification we need to make in this step are really simple.	

```
DEBUT();

MA=DEFI_MATERIAU(ELAS=_F(E=200000.0,
                         NU=0.3,),);

MAIL=LIRE_MAILLAGE(FORMAT='MED',);

MAIL=MODI_MAILLAGE(reuse =MAIL,
                   MAILLAGE=MAIL,
                   ORIE_PEAU_3D=_F(GROUP_MA='Load',),);

MODE=AFFE_MODELE(MAILLAGE=MAIL,
                 AFFE=_F(TOUT='OUI',
                         PHENOMENE='MECANIQUE',
                         MODELISATION='3D',),);

MATE=AFFE_MATERIAU(MAILLAGE=MAIL,
                   AFFE=_F(TOUT='OUI',
                           MATER=MA,),);

CHAR=AFFE_CHAR_MECA(MODELE=MODE,
                    DDL_IMPO=(_F(GROUP_MA='Fix',
                                DX=0.0,
                                DY=0.0,
                                DZ=0.0,),
                                ),
                    PRES_REP=(_F(GROUP_MA='Load',
                              PRES=1.0,), ),);

RESU=MECA_STATIQUE(MODELE=MODE,
                   CHAM_MATER=MATE,
                   EXCIT=_F(CHARGE=CHAR,),);

RESU=CALC_ELEM(reuse =RESU,
               MODELE=MODE,
               CHAM_MATER=MATE,
               RESULTAT=RESU,
               OPTION=('SIGM_ELNO','SIEQ_ELNO',),
               EXCIT=_F(CHARGE=CHAR,),);
```

#	Description	Figure
3.	Find the sentence with the words "*PRES_REP*" which is the pressure term that we entered on face "*Load*" with a value of "*1*" (default). Select all the lines that relate to that command and delete them by pressing Delete.	
4.	Replace the above sentence with the one shown in the figure on the right. "*FORCE_NODALE*" means Nodal Force in French. Here we replaced *PRES_REP* which may also be a key word to apply loads BUT it didn't give us the option to apply that force in the direction that we were interested in. The amount of Load that we want to be applied is 500N and it should have a direction of Negative Y (-Y). FORCE_NODALE → Nodal Force. GROUP_NO → In a specific group of Nodes (Load). FY → Direction.	
5.	Save the file with our new modifications.	

3.

```
DEBUT();

MA=DEFI_MATERIAU(ELAS=_F(E=200000.0,
                         NU=0.3,),);

MAIL=LIRE_MAILLAGE(FORMAT='MED',);

MAIL=MODI_MAILLAGE(reuse =MAIL,
                   MAILLAGE=MAIL,
                   ORIE_PEAU_3D=_F(GROUP_MA='Load',)

MODE=AFFE_MODELE(MAILLAGE=MAIL,
                 AFFE=_F(TOUT='OUI',
                         PHENOMENE='MECANIQUE',
                         MODELISATION='3D',),);

MATE=AFFE_MATERIAU(MAILLAGE=MAIL,
                   AFFE=_F(TOUT='OUI',
                           MATER=MA,),);

CHAR=AFFE_CHAR_MECA(MODELE=MODE,
                    DDL_IMPO=(_F(GROUP_MA='Fix',
                                DX=0.0,
                                DY=0.0,
                                DZ=0.0,),
                                ),
                    PRES_REP=(_F(GROUP_MA='Load',
                              PRES=1.0,), ),);

RESU=MECA_STATIQUE(MODELE=MODE,
                   CHAM_MATER=MATE,
                   EXCIT=_F(CHARGE=CHAR,),);

RESU=CALC_ELEM(reuse =RESU,
               MODELE=MODE,
               CHAM_MATER=MATE,
```

4.

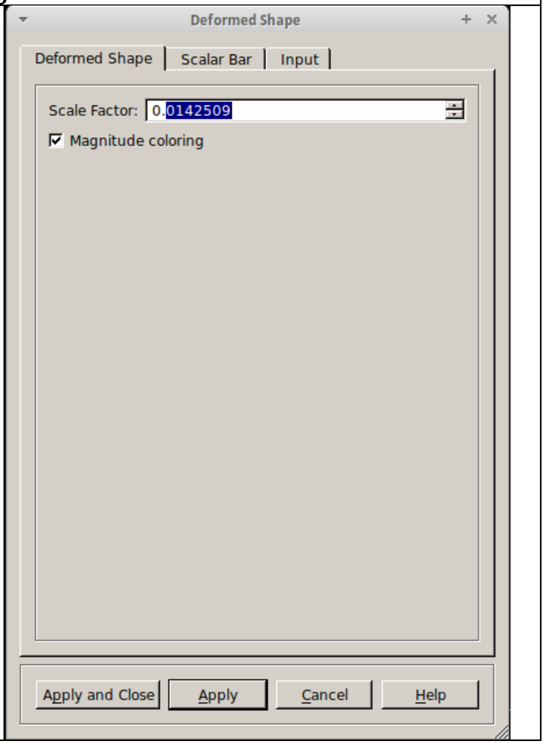
```
DEBUT();

MA=DEFI_MATERIAU(ELAS=_F(E=200000.0,
                         NU=0.3,),);

MAIL=LIRE_MAILLAGE(FORMAT='MED',);

MAIL=MODI_MAILLAGE(reuse =MAIL,
                   MAILLAGE=MAIL,
                   ORIE_PEAU_3D=_F(GROUP_MA='Load',),);

MODE=AFFE_MODELE(MAILLAGE=MAIL,
                 AFFE=_F(TOUT='OUI',
                         PHENOMENE='MECANIQUE',
                         MODELISATION='3D',),);

MATE=AFFE_MATERIAU(MAILLAGE=MAIL,
                   AFFE=_F(TOUT='OUI',
                           MATER=MA,),);

CHAR=AFFE_CHAR_MECA(MODELE=MODE,
                    DDL_IMPO=_F(GROUP_MA='Fix',
                                DX=0.0,
                                DY=0.0,
                                DZ=0.0,),
                    FORCE_NODALE=_F(GROUP_NO='Load',
                                    FY=-500,),);

RESU=MECA_STATIQUE(MODELE=MODE,
                   CHAM_MATER=MATE,
                   EXCIT=_F(CHARGE=CHAR,),);

RESU=CALC_ELEM(reuse =RESU,
               MODELE=MODE,
               CHAM_MATER=MATE,
               RESULTAT=RESU,
               OPTION=('SIGM_ELNO','SIEQ_ELNO',),
               EXCIT=_F(CHARGE=CHAR,),);
```

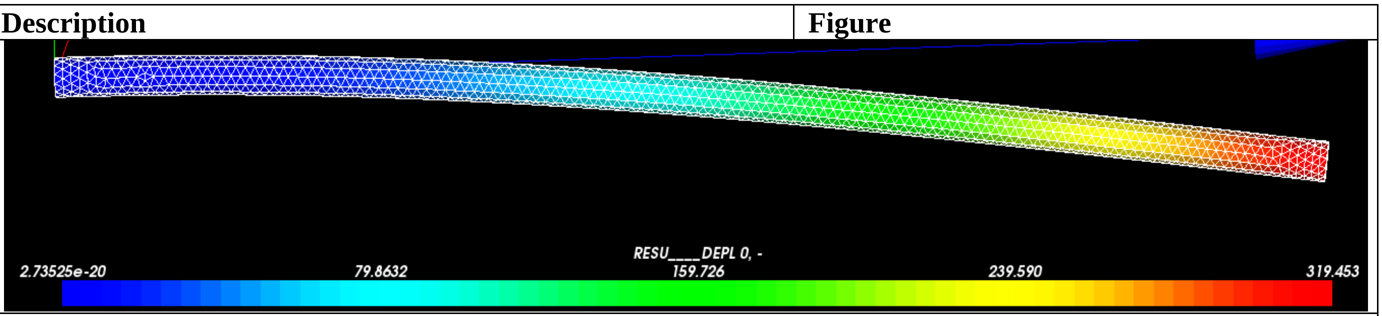
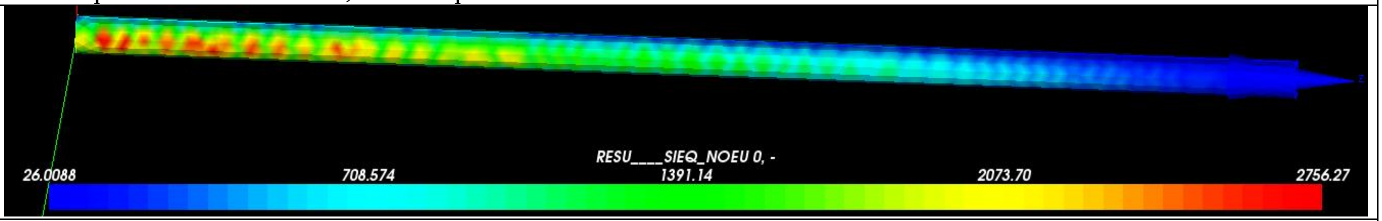
Step 10: Running the Analysis

#	Description	Figure
1.	Open Salome_Meca again and check that your .hdf file is open. Make sure that Aster Module is selected. Right-click to the "*linear-static*" branch in the Object Browser and click "*Run*". The pop-up window where the white letters and numbers flow will appear. If everything goes well, you will be able to see "*Post-Pro*" Module appear in the Object Browser.	
2.	Most of the times that you run an analysis the "*Post-Pro*" branch will not appear in the Object Browser. That happens when there is a code error/mistake. To fix those mistakes you will have to go back to the command file, make your changes and then switch back to Salome_Meca, right-click on "*linear-static*" again, choose "*Edit*" and instantly click "*Ok*", without changing anything inside that window. This way the analysis will get updated with the changes you made in the command file. Right-click and "*Run*" to compute the new data. There is a way that points out where exactly in your code appears to be the error, but we will address to that in the next chapter.	
3.	Save your work.	

Step 11: Post-Processing of the results

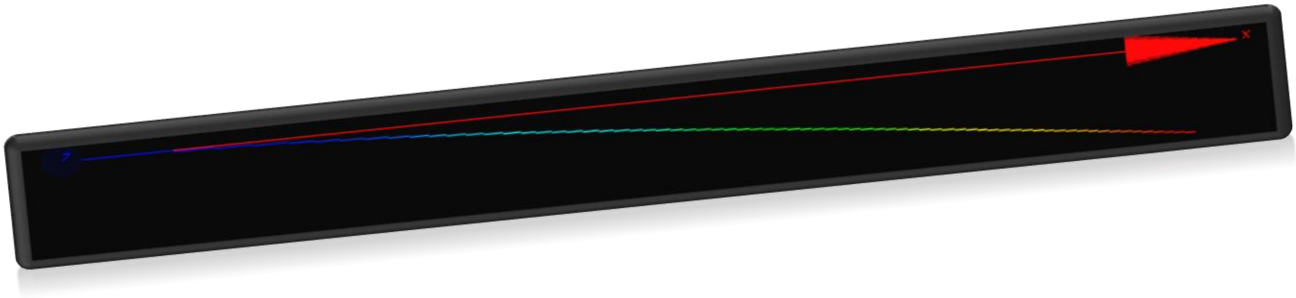
# Since you are familiar already with the way that Post-Pro Module visualizes the results, we will continue together but without thorough explanation.

#	Description	Figure
1.	Activate "*Post-Pro*" Module from the drop down list. Open "*RESU___DEPL*" branch and add "*Deformed Shape*". To make the deformation more prominent, change the scale factor to "*0.3*". *Finish with "Apply and Close"* For this Deformed Shape, enable "*Show Only*" and change the Representation to "*SurfaceFrame*".	 Deformed Shape Deformed Shape Scalar Bar Input Apply and Close Apply Cancel Help Scale Factor: 0.0142509 ☑ Magnitude coloring

#	Description	Figure
2.	 RESU____DEPL 0, - 2.73525e-20 79.8632 159.726 239.590 319.453	
3.	Open “*RESU___SIEQ_NOEU*” branch and add Scalar Map with the mode “*VMIS*”. For the VonMises Stresses, also enable “*Show Only*” and change the representation to “*Surface*”. Min. Equiv. Stress = 26 MPa, Max. Equiv. Stress = 2756.3 MPa.	
4.	 RESU____SIEQ_NOEU 0, - 26.0088 708.574 1391.14 2073.70 2756.27	
5.	Don’t mind the results that are shown in the last two figures. The purpose of this analysis was to show you how to manually edit the command (.comm) file.	

Chapter_III: Child Swing

In this chapter we will examine the 1D_Analysis of a child swing beam.
Emphasizing in Nominal Stress.



Step 1: Purpose of the FE Analysis/ Description of the problem

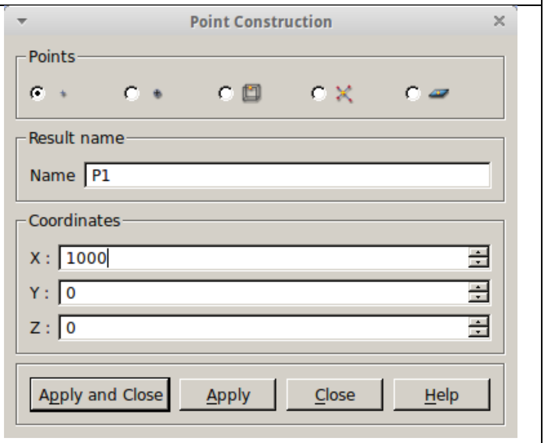
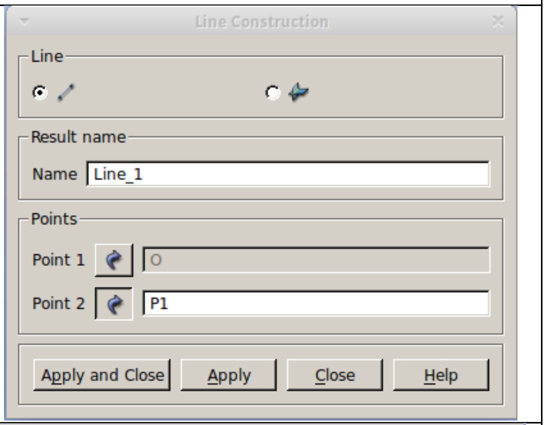
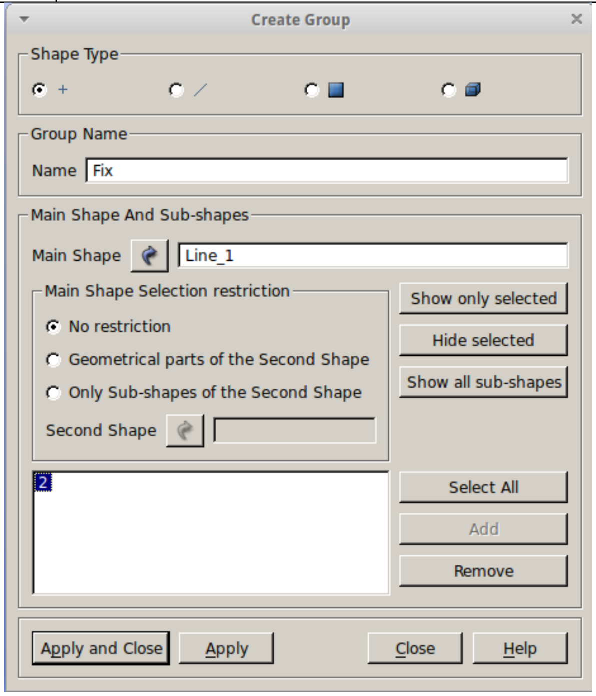
The purpose of this chapter is to find out the deformation of the beam, which consist the child swing, and the displacement of its end point. Also, we will examine how the stress will look like, when a loading by a specific weight, defined by the force.

Step 2: Input Values for the FE Analysis

Beam Dimensions:	l= 1000mm;
Cross-Section:	r=15mm;
Young’s Modulus:	2e5 MPa;
Poisson’s Ratio:	0.3;
Pressure Applied on the “right edge” face:	FY= -500N;

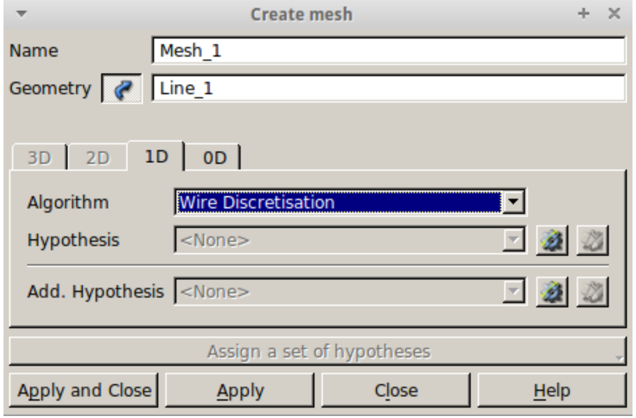
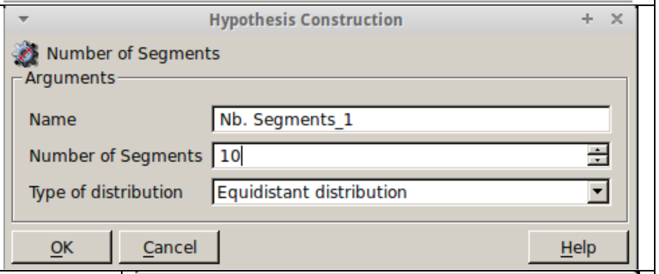
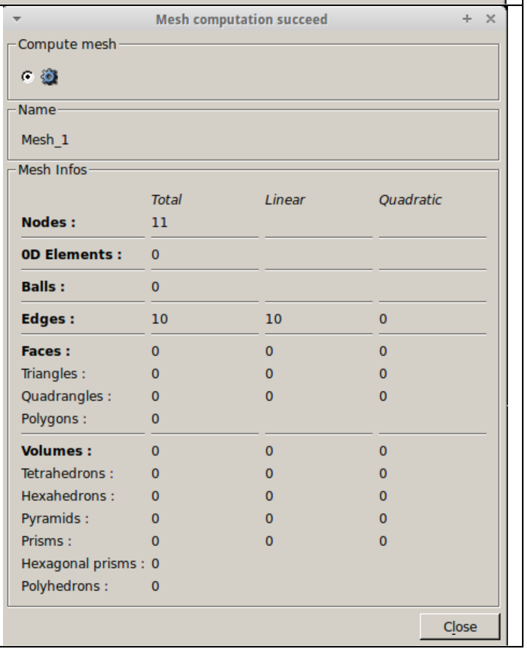
Step 3: Model Geometry

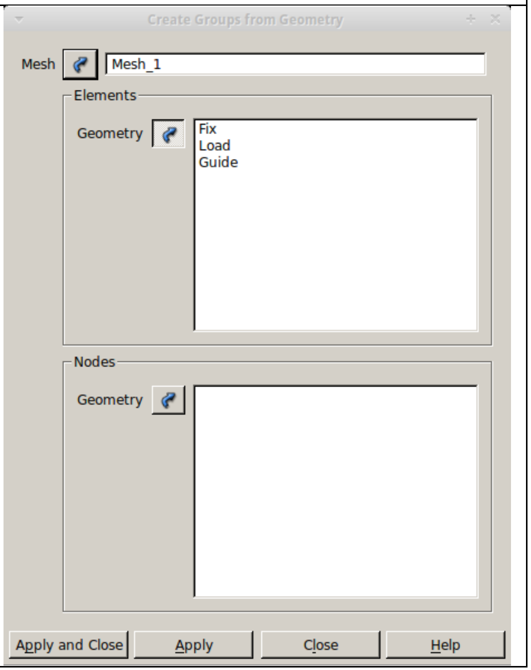
# Open Salome_Meca, start a new study, enable Geometry Module and save your study with an appropriate name.

#	Description	Figure
1.	Click New Entity → Basic → Point. A pop-up window appears like the figure on the right. Select the 1st Algorithm in Points. Enter "*P1*" as a name. Enter (1000,0,0) for X,Y,Z coordinates. *Finish with "Apply and Close"*	 Name: P1 X : 1000 Y : 0 Z : 0 Point Construction Points Result name Coordinates Apply and Close Apply Close Help
2.	Now we need to create a line with 2 points (0,0) and P1. Click New Entity → Basic → Line. Select 1st Algorithm for line by two points. Click on the blue curved arrow near Point 1, to choose our first point of the line and repeat the same for Point 2, as shown in the figure on the right. *Finish with "Apply and Close"*	 Name: Line_1 Point 1: O Point 2: P1 Line Construction Line Result name Points Apply and Close Apply Close Help
3.	Next phase is to arrange our groups, like we did in the previous chapter. Click on New Entity → Group → Create. In the pop-up window we will follow the same steps we did before, BUT this time we will select the 1st Algorithm for selecting the points, which is located at the beginning of our line. *Continue with "Apply"* Do the same for the other end and name it "*Load*". *Continue with "Apply"* Now to create a group for our main line, we will need to select the 2nd Algorithm for Line, and name it "*Guide*". Select the line in the viewer tab and click "*Add*". *Finish with "Apply and Close"*	 Name: Fix Main Shape: Line_1 ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape 2 Create Group Shape Type Group Name Main Shape And Sub-shapes Main Shape Selection restriction Second Shape Show only selected Hide selected Show all sub-shapes Select All Add Remove Apply and Close Apply Close Help
4.	Save the file and give it a proper name.	

Step 4: Meshing Geometry

# After we are done with modeling our geometry it is time to move to Mesh Module. Enable Mesh Module in Salome_Meca and start the procedure to generate the Mesh.

#	Description	Figure
1.	Click Mesh → Generate Mesh. Select Line_1 for Geometry. Select "*Wire Discretisation*" for Algorithm from the drop down list. Click on the gear button next to Hypothesis.	Create mesh Name: Mesh_1 Geometry: Line_1 3D 2D 1D 0D Algorithm: Wire Discretisation Hypothesis: <None> Add. Hypothesis: <None> Assign a set of hypotheses Apply and Close / Apply / Close / Help
2.	In the list that will fall down, select "*Nb.Segments*", keep default name and change the number of segments to "10". *Click "Ok".* *Finish with "Apply and Close"*	Hypothesis Construction Number of Segments Arguments Name: Nb. Segments_1 Number of Segments: 10 Type of distribution: Equidistant distribution OK / Cancel / Help
3.	Right-click on "*Mesh_1*" from the Object Browser and click "*Compute*". If everything goes well you should have the same pop-up window as shown in the figure.	Mesh computation succeed Compute mesh Name: Mesh_1 Mesh Infos (Total / Linear / Quadratic) Nodes : 11 0D Elements : 0 Balls : 0 Edges : 10 / 10 / 0 Faces : 0 / 0 / 0 Triangles : 0 / 0 / 0 Quadrangles : 0 / 0 / 0 Polygons : 0 Volumes : 0 / 0 / 0 Tetrahedrons : 0 / 0 / 0 Hexahedrons : 0 / 0 / 0 Pyramids : 0 / 0 / 0 Prisms : 0 / 0 / 0 Hexagonal prisms : 0 Polyhedrons : 0 Close

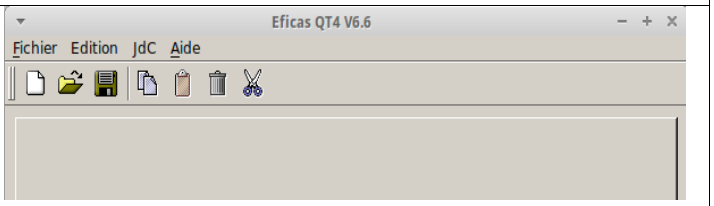
#	Description	Figure
4.	Right-click on “*Mesh_1*” again and choose “*Create Groups From Geometry*”. Then you have to select the groups we created in Geometry Module and place them in the same box as in the figure on the right. *Finish with “Apply and Close”*	 Mesh: Mesh_1 Elements – Geometry: Fix, Load, Guide Nodes – Geometry: Create Groups from Geometry Apply and Close Apply Close Help

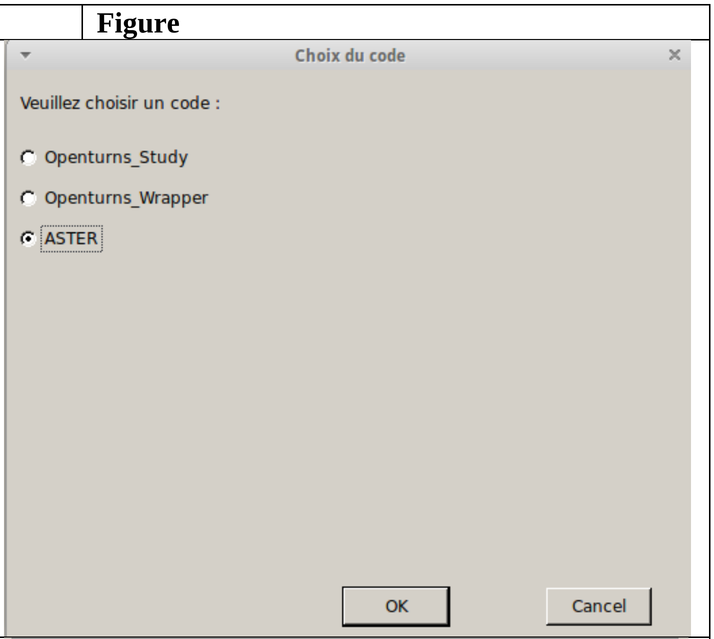
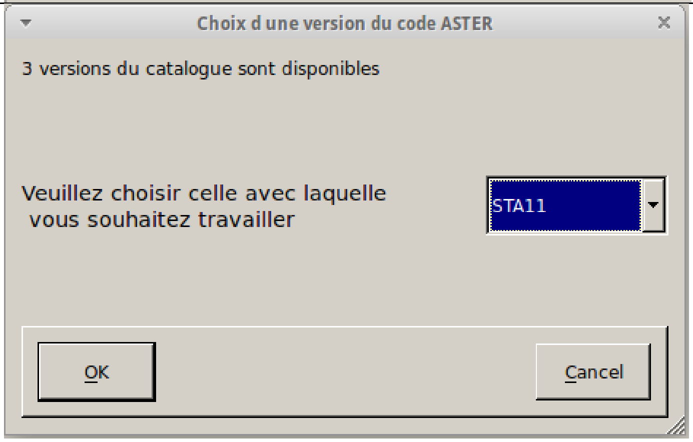
# We are done meshing our geometry. Next step is creating the command file that will contain all the necessary information and will give instructions to Aster about the analysis. This time we are going to use Eficas to conclude our command file.

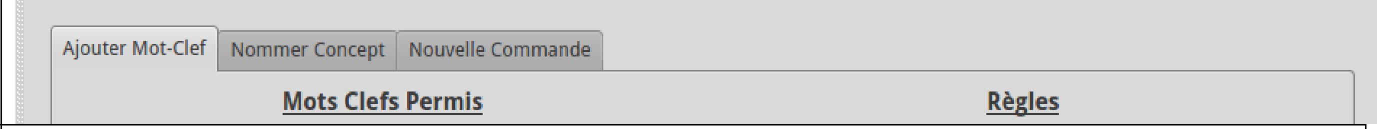
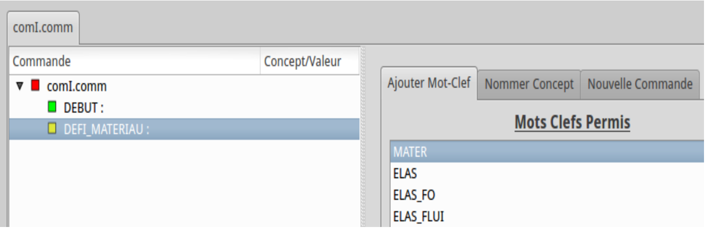
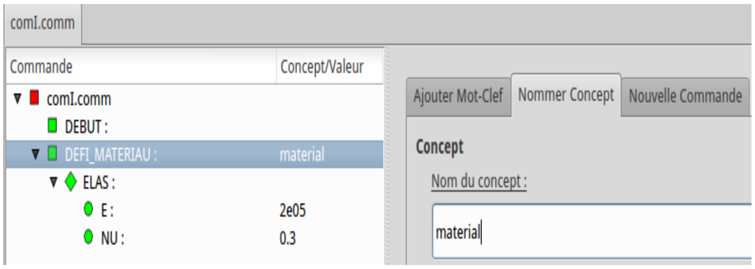
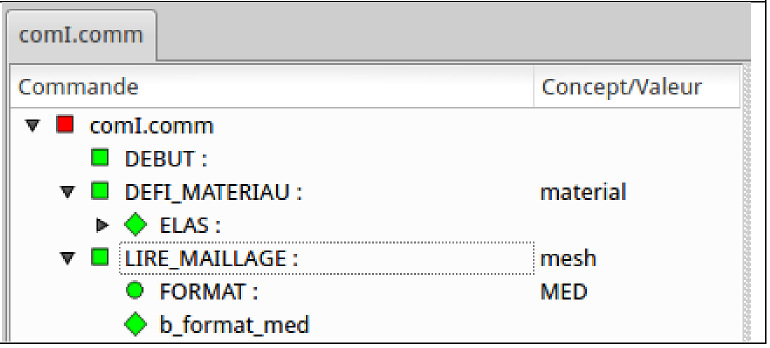
Step 5,6,7,8 and 9: Creating command (.comm) file with Eficas

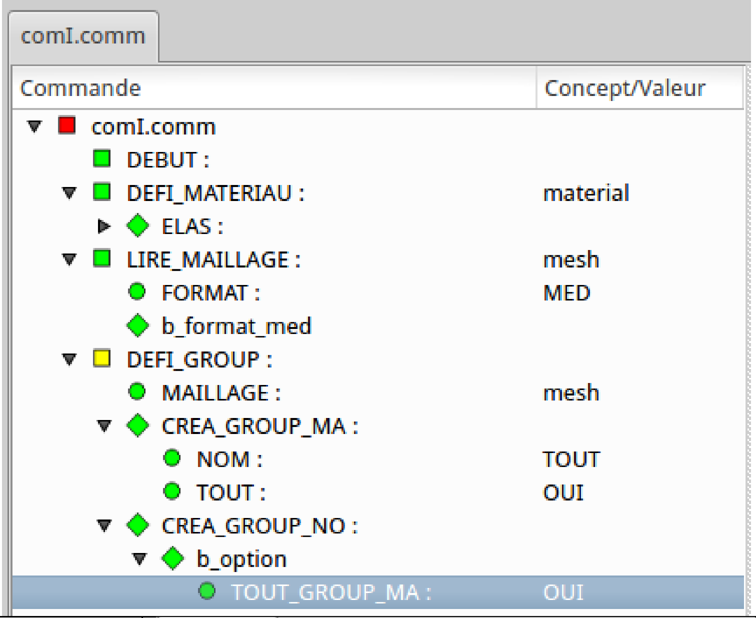
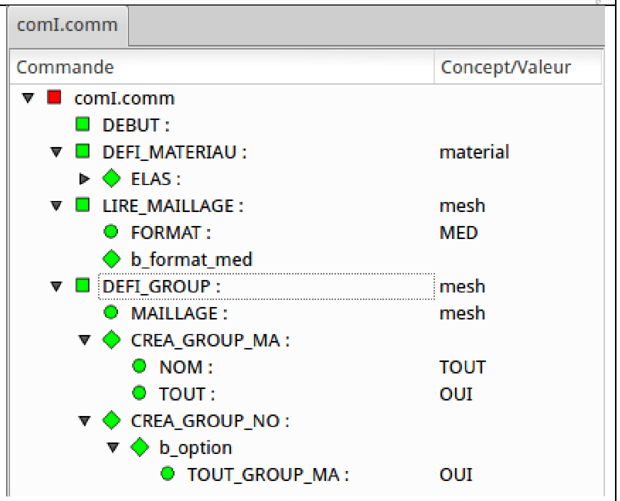
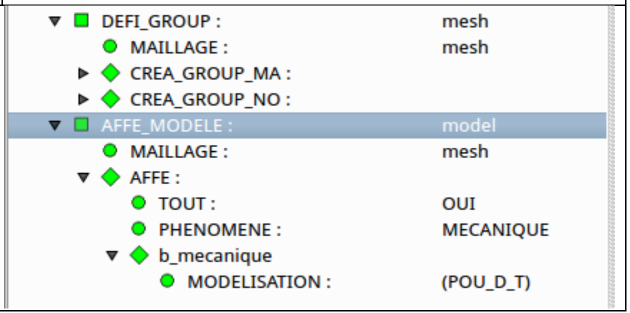
# As we noticed from the previous chapter there is more than one way to create a valid command file. In the previous chapter we used Salome_Meca Wizard and then we modified it in a text editor. In this chapter we will use Eficas Application.

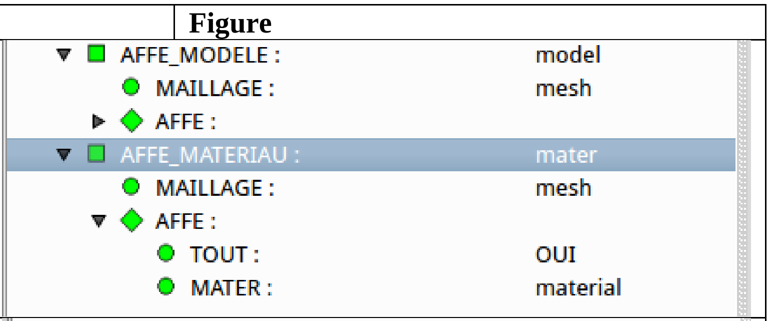
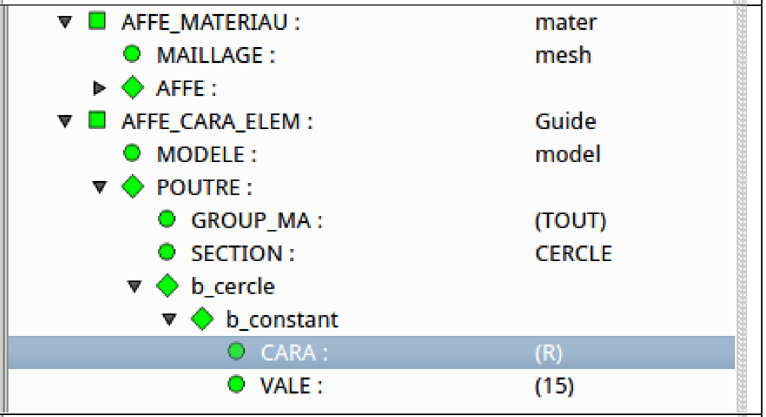
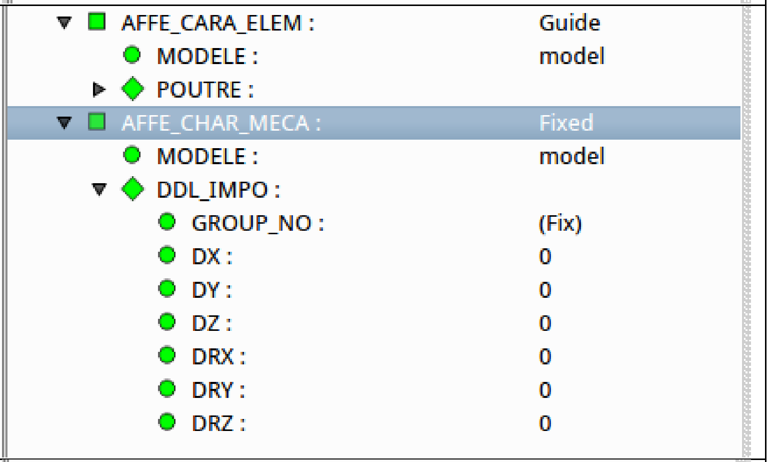
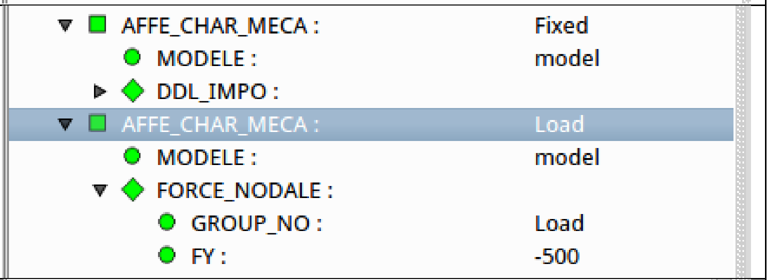
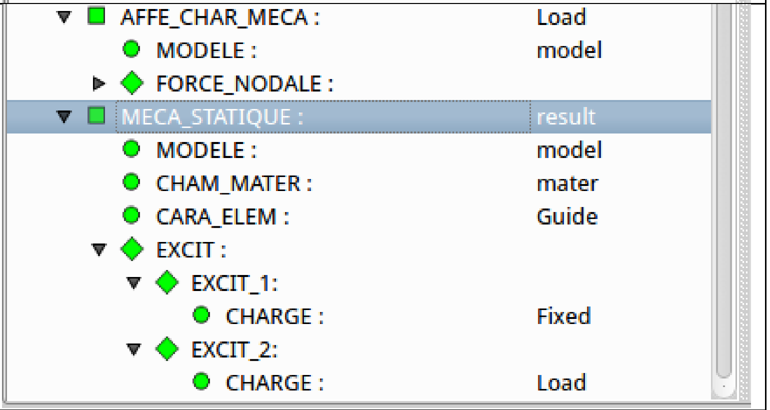
# Eficas is an application that generates syntactically correct command files, this does not mean that they will make sense or that there will not be any errors while running the analysis. This applications only makes certain that you won’t miss any commas or any closing brackets, all other features have to come from the user.

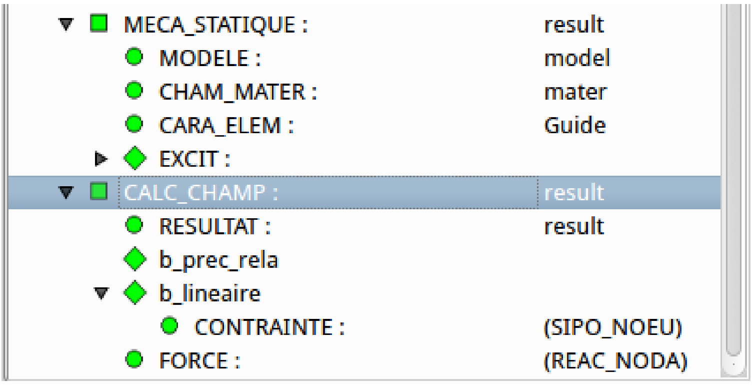
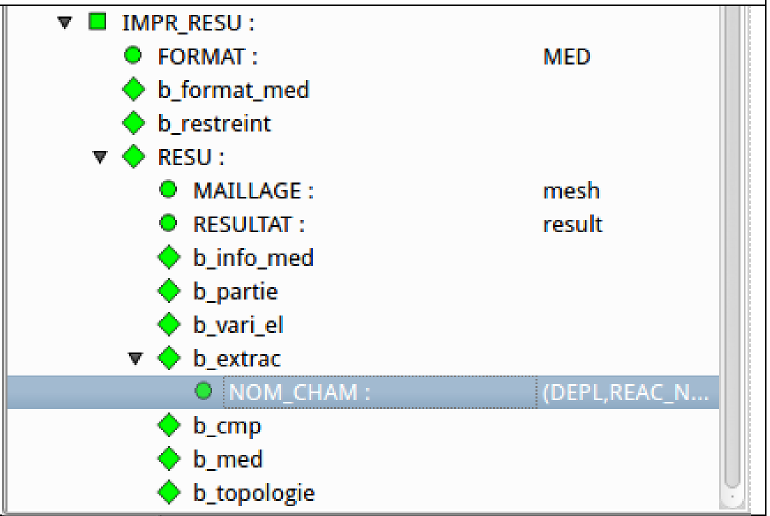
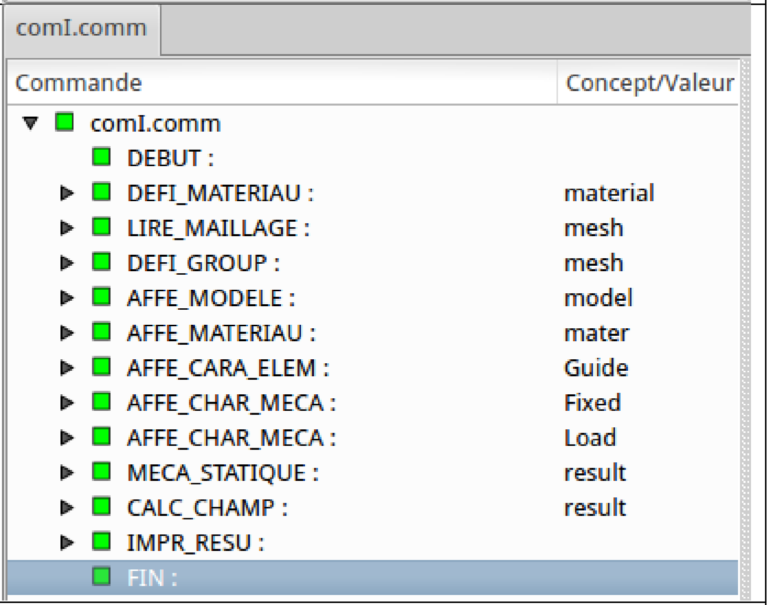
#	Description	Figure
1.	In Salome_Meca select “*Eficas*” from the drop down list. A huge discomfort will be that eficas has not yet been translated to English and everything, including the tool tips are in French language.	
2.	In the main Toolbar, click on Eficas → Eficas Multicatalogue. Icons from left to right: 1] Create new file, 2] Open file, 3] Save file, 4] Copy, 5] Paste, 6] Delete, 7] Cut. Click on Create a new file.	

#	Description	Figure
3.	In the first pop-up window, you will be asked to choose a code. Select “*ASTER*”. *Continue with “Ok”*	 Choix du code Veuillez choisir un code : ○ Openturns_Study ○ Openturns_Wrapper ● ASTER
4.	In the second pop-up window, you will be asked to choose in which version of Code_Aster you want to work on. Select “*STA11*” which is the newest one (at my system) and the most stable. *Continue with “Ok”*	 Choix d une version du code ASTER 3 versions du catalogue sont disponibles Veuillez choisir celle avec laquelle vous souhaitez travailler: STA11
5.	Before going any further, let’s save the file in the same folder we saved the study before. Click on Fishier → En registrer sous (Save us) and click “*Save*”.	
6.	In the figure below you can see how Eficas application looks before we start working in it. In the left column it shows the file name “ComI.comm” in red colour which means that the file is not yet complete. Eficas is waiting for us to input the data, when that happens correctly the red square will turn green, meaning that our command file is correct. In the second column there are all the commands that are available in Code_Aster and are sorted Alphabetically. The data that needs to be added now, is called concepts, and for a fully functionable command file we need to add the following concepts: i. DEBUT [Start] ii. Material Definition […] iii. What type of Mesh to read […] iv. Type of modelling [1D,2D,3D,Mechanical] v. Assign previously defined Material to the model […] vi. Add Geometric Boundary Conditions […] vii. Add Load Boundary Conditions […] viii. Define Type of Analysis […] ix. Results to be calculated at the elements, nodes […] x. Save the results to a MED file […] xi. Fin [End]	

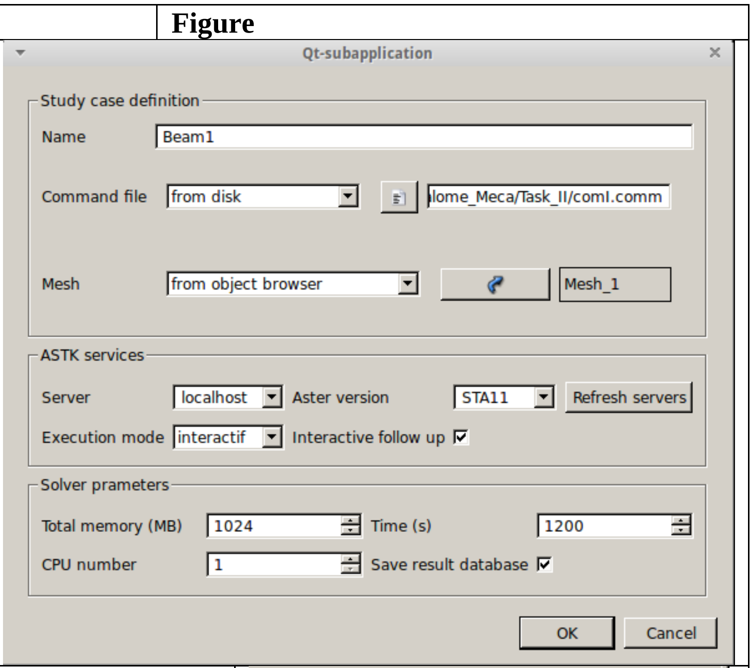
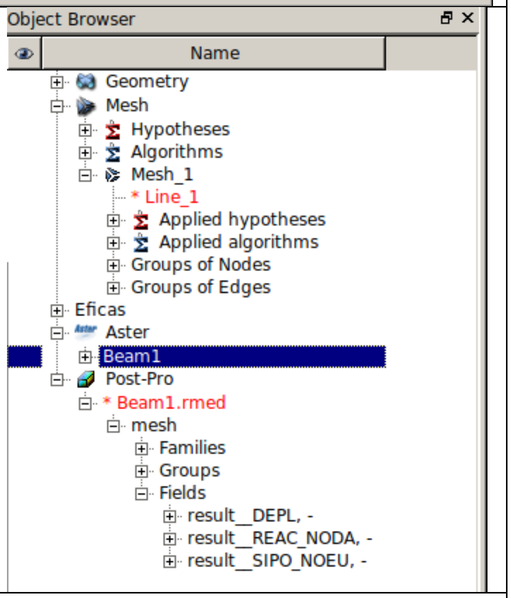
#	Description	Figure
7.	Ajouter Mot-Clef = Add Keyword Nommer Concept = Name of Concept Nouvelle Commande = New Command *Note: If you are having trouble with undestanding the meaning of some words/commands remember to use translation from French.*	Mots Clefs Permis = Permitted Keywords Regles = Rules
8.	 Ajouter Mot-Clef Nommer Concept Nouvelle Commande Mots Clefs Permis Règles	
9.	Find "*DEBUT*"command from the second column, double-click on it to begin our command file. You can see that "*DEBUT*" square is green, which mean that this command is functionable and we can move to the next command. Clicking on the command that you just added, will get you back to the first screen where you can select your next type of command/concept.	
10.	Second step will be defining the material that we are interested in using. Choose "*Nouvelle Commande*" tab, to select a new concept, and find "*DEFI_MATERIAU*" which means DEFINE_MATERIAL. Double-click or select it and click "*Valider*" Now you can see that DEFI_MATERIAU has been added to our command list but the square is yellow, which means that we need to add more information to make this command functionable.	 comI.comm Commande Concept/Valeur comI.comm DEBUT : DEFI_MATERIAU : Ajouter Mot-Clef Nommer Concept Nouvelle Commande Mots Clefs Permis MATER ELAS ELAS_FO ELAS_FLUI
11.	Find "*ELAS*" from "*Mots Clefs Permis*" and double-click on it to add it to the command file. Two other branched will appear, fill them like in the figure on the right. "*E*" stands for Young's Modulus and "*NU*" for Poison's Ratio. After this step you will notice in the command file list that every square is green except DEFI_MATERIAU, and that happens because we need to assign a name to the concept from "*Nommer Concept*" tab.	 comI.comm Commande Concept/Valeur Ajouter Mot-Clef Nommer Concept Nouvelle Commande Concept comI.comm DEBUT : DEFI_MATERIAU : material ELAS : E : 2e05 NU : 0.3 Nom du concept : material
12.	In the next step we must define the type of the mesh that will be read as input. Go to "*Nouvelle Command*" again and choose "*LIRE_MAILLAGE*" → FORMAT=MED and lastly give the command a proper name.	 comI.comm DEBUT : DEFI_MATERIAU : material ELAS : LIRE_MAILLAGE : mesh FORMAT : MED b_format_med

#	Description	Figure
13.	We came to the point where we will need to apply Boundary Conditions on the nodes, but we can't do that unless the MED file has information for the node group. The DEFI_GROUP command will help us do that. Click on Nouvelle Commande → DEFI_GROUP → MAILLAGE → mesh. Click on DEFI_GROUP and double-click CREA_GROUP_MA, click on NOM and name it "*TOUT*" and click "*Valider*". Click on TOUT → OUI which means give the name (NOM) TOUT (all) to all mesh elements. Here we made a group of the entire mesh and called it "*TOUT*".	 comI.comm Commande | Concept/Valeur comI.comm DEBUT : DEFI_MATERIAU : material ELAS : LIRE_MAILLAGE : mesh FORMAT : MED b_format_med DEFI_GROUP : MAILLAGE : mesh CREA_GROUP_MA : NOM : TOUT TOUT : OUI CREA_GROUP_NO : b_option TOUT_GROUP_MA : OUI
14.	Now we need to convert this group of mesh to a group of nodes. Click DEFI_GROUP → CREA_GROUP_NO → b_option → TOUT_GROUP_MA → OUI, which will convert all mesh group to Node groups. *Name the concept and move to the next one.*	 comI.comm Commande | Concept/Valeur comI.comm DEBUT : DEFI_MATERIAU : material ELAS : LIRE_MAILLAGE : mesh FORMAT : MED b_format_med DEFI_GROUP : mesh MAILLAGE : mesh CREA_GROUP_MA : NOM : TOUT TOUT : OUI CREA_GROUP_NO : b_option TOUT_GROUP_MA : OUI
15.	For the next step we need to define the type of modelling. "*AFFE_MODELE*" command is the most suitable command for that task. Choose AFFE_MODELE → MAILLAGE → mesh. Select TOUT → OUI, to asign the modeling to the whole mesh. Select PHENOMENE → MECANIQUE Select MODELISATION → POU_D_T *Close the command with naming it.*	 DEFI_GROUP : mesh MAILLAGE : mesh CREA_GROUP_MA : CREA_GROUP_NO : AFFE_MODELE : model MAILLAGE : mesh AFFE : TOUT : OUI PHENOMENE : MECANIQUE b_mecanique MODELISATION : (POU_D_T)

#	Description	Figure
16.	Next step will be to Add Material Properties which were defined previously to the model. Choose AFFE_MATERIAU (Assign Material) → MAILLAGE → mesh. Click AFFE → TOUT → OUI MATER → material (depends of the given name). *Finish with naming the concept.*	 AFFE_MODELE : model MAILLAGE : mesh AFFE : AFFE_MATERIAU : mater MAILLAGE : mesh AFFE : TOUT : OUI MATER : material
17.	Next step is to define the Geometry of the beam. Choose AFFE_CARA_ELEM and fill the gaps according to the figure on the right. Here we defined a Circle Cross-Section to our beam, with R=15mm. *Finish with naming the concept.*	 AFFE_MATERIAU : mater MAILLAGE : mesh AFFE : AFFE_CARA_ELEM : Guide MODELE : model POUTRE : GROUP_MA : (TOUT) SECTION : CERCLE b_cercle b_constant CARA : (R) VALE : (15)
18.	In the next step we will Add Geometric Boundary Conditions. Select AFFE_CHAR_MECA and fill in the necessary information as shown in the figure on the right. According to our data, we want 0 value to all degrees of freedom. *Finish with naming the concept.*	 AFFE_CARA_ELEM : Guide MODELE : model POUTRE : AFFE_CHAR_MECA : Fixed MODELE : model DDL_IMPO : GROUP_NO : (Fix) DX : 0 DY : 0 DZ : 0 DRX : 0 DRY : 0 DRZ : 0
19.	In the next step we will Add Load Boundary Conditions. Select AFFE_CHAR_MECA again and give in the necessary information as shown in the figure on the right. Here we added a load of 500N to the "*Load*" group of nodes with negative FY direction. *Finish with naming the concept.*	 AFFE_CHAR_MECA : Fixed MODELE : model DDL_IMPO : AFFE_CHAR_MECA : Load MODELE : model FORCE_NODALE : GROUP_NO : Load FY : -500
20.	Next step is defining the type of analysis. Add MECA_STATIQUE (Static Mechanical Analysis) to the command file and then add information to the branch as shown in the figure on the right. Note that we are adding two EXCIT command to cater for two types of Boundary Conditions (EXCIT = excitation of the calculation). *Finish with naming the concept.*	 AFFE_CHAR_MECA : Load MODELE : model FORCE_NODALE : MECA_STATIQUE : result MODELE : model CHAM_MATER : mater CARA_ELEM : Guide EXCIT : EXCIT_1: CHARGE : Fixed EXCIT_2: CHARGE : Load

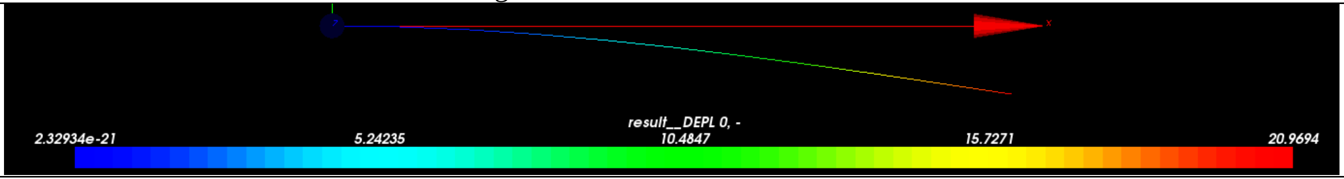
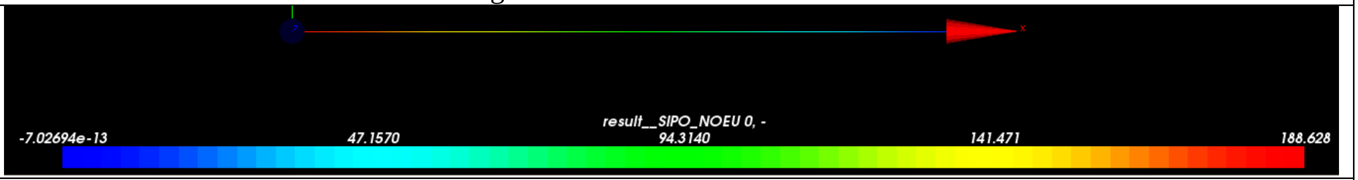
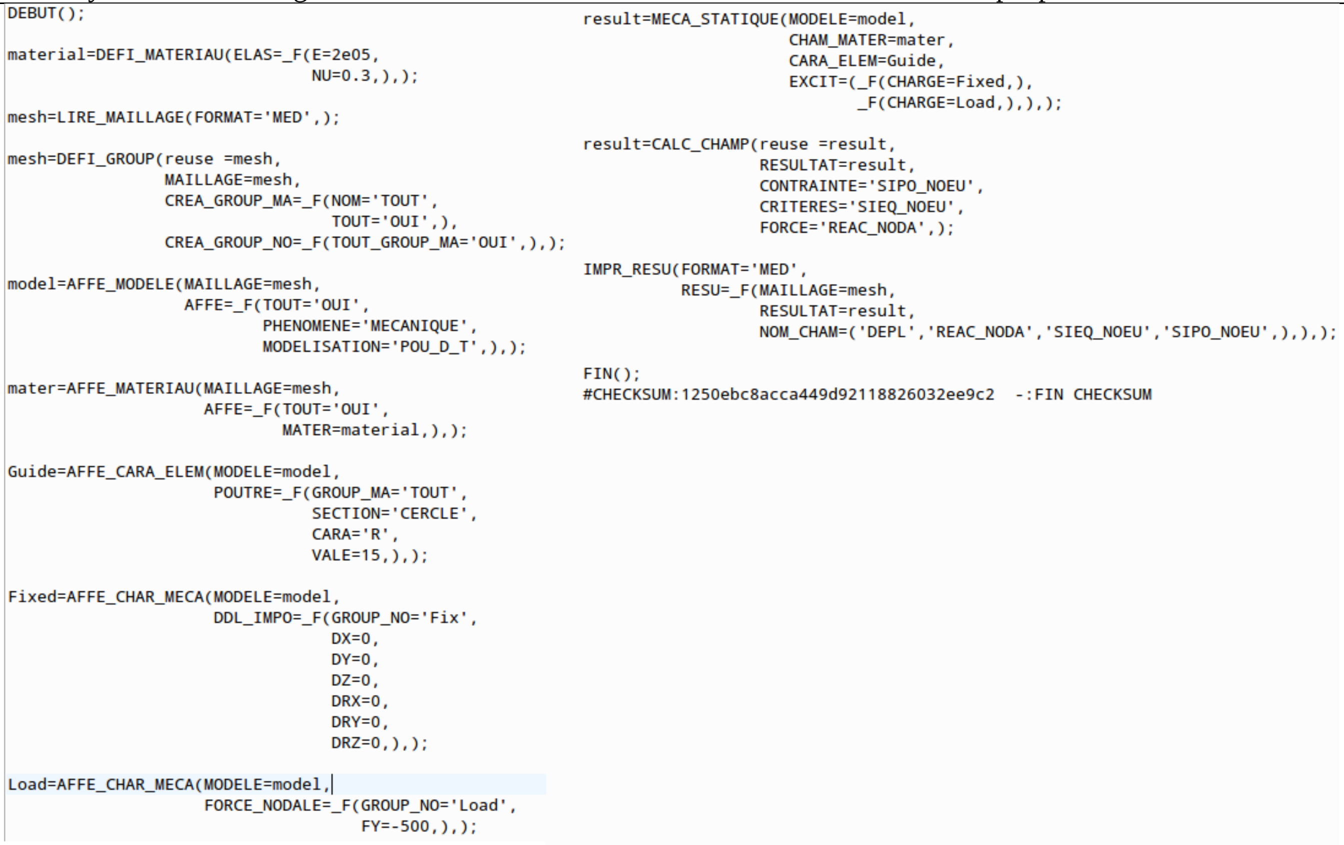
#	Description	Figure
21.	Next we define the types of result to be calculated from the initial displacement calculated. Add CALC_CHAMP to the command file and fill the information according to the figure on the right. Name it with the same name you used with the previous concept. Here we want results for 2 information calculated. SIPO_NOEU = Stress at Nodes. REAC_NODA = Reaction at Nodes.	 MECA_STATIQUE : result MODELE : model CHAM_MATER : mater CARA_ELEM : Guide EXCIT : CALC_CHAMP : result RESULTAT : result b_prec_rela b_lineaire CONTRAINTE : (SIPO_NOEU) FORCE : (REAC_NODA)
22.	In this step we need to define the information that will be written in the MED file. Add IMPR_RESU and follow the way the figure shows. At this point our goal is to print and visualize some results. DEPL for deflection and the other 2 that we explained in the previous step #21.	 IMPR_RESU : FORMAT : MED b_format_med b_restreint RESU : MAILLAGE : mesh RESULTAT : result b_info_med b_partie b_vari_el b_extrac NOM_CHAM : (DEPL,REAC_N... b_cmp b_med b_topologie
23.	Last step will be adding the FIN command. Once you add FIN, "*comI.comm*" square turns green, meaning that our command file is completed.	 comI.comm Commande Concept/Valeur comI.comm DEBUT : DEFI_MATERIAU : material LIRE_MAILLAGE : mesh DEFI_GROUP : mesh AFFE_MODELE : model AFFE_MATERIAU : mater AFFE_CARA_ELEM : Guide AFFE_CHAR_MECA : Fixed AFFE_CHAR_MECA : Load MECA_STATIQUE : result CALC_CHAMP : result IMPR_RESU : FIN :
27.	Save this file and return to Salome_Meca.	

Step 10: Run the Analysis

#	Description	Figure
1.	Now that we created our command file with the help of Eficas Application, we need to define a study case in Aster that will do the analysis and point the location of our command file. Choose Aster Module from the drop down list. Click Aster → Add Study Case. Chose a correct name. Locate your command (.comm) file from your system. Set Mesh_1 to the Mesh. And input the Solver Prameters as shown. *Finish with "Ok"*.	 Qt-subapplication Study case definition Name: Beam1 Command file: from disk, lome_Meca/Task_II/comI.comm Mesh: from object browser, Mesh_1 ASTK services Server: localhost; Aster version: STA11; Refresh servers Execution mode: interactif; Interactive follow up ☑ Solver prameters Total memory (MB): 1024; Time (s): 1200 CPU number: 1; Save result database ☑ OK Cancel
2.	When you click "*Ok*", a new branch appears to the Object Browser named Aster, which contains the study case that we just created. Choose and right-click to "*Run*" the analysis. If everything is correct, after the analysis is complete you should be able to see "*Post-Pro*" branch added to the Object Browser, like in the figure on the right.	 Object Browser Name Geometry Mesh Hypotheses Algorithms Mesh_1 * Line_1 Applied hypotheses Applied algorithms Groups of Nodes Groups of Edges Eficas Aster Beam1 Post-Pro * Beam1.rmed mesh Families Groups Fields result__DEPL, - result__REAC_NODA, - result__SIPO_NOEU, -
3.	Save the study and let's have a look at the results.	

Step 11: Post Processing the results

# Open Post-Pro Module from the drop down list and let’s see what the FE Analysis has to show.

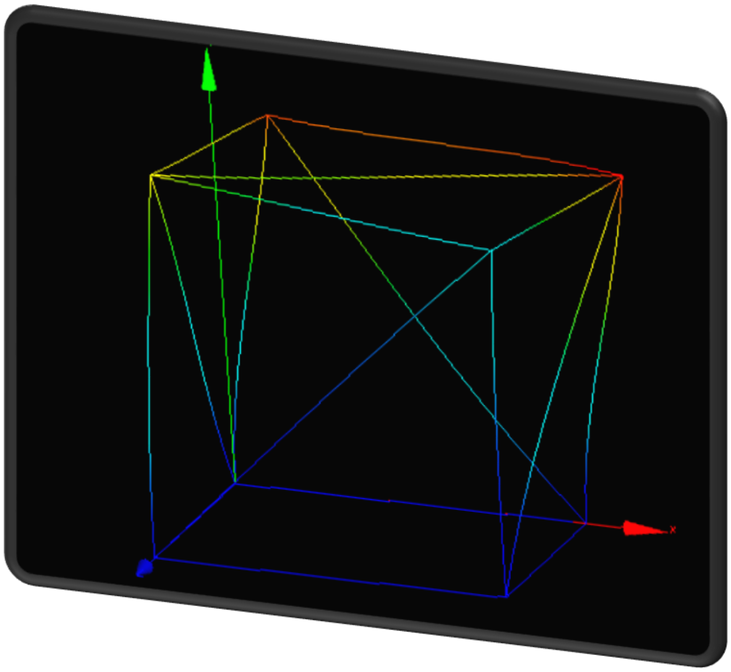
#	Description	Figure
1.	Follow the same steps we took in the previous chapter. Result_DEPL → Deformed Shape → Show Only → Surface. We can see that the deformation has a range from 2.32934e-21 mm to 20.9694 mm	
2.	 result__DEPL 0, - 2.32934e-21 5.24235 10.4847 15.7271 20.9694	
3.	Next we will need to see the stress results. Result_SIPO_NOEU → Scalar Map → Mode: SMFZ (Stress due to bending Moment MZ) → Show Only. We can see that the direct stress has a range from -7.02694e-13 MPa to 188.628 MPa.	
4.	 result__SIPO_NOEU 0, - -7.02694e-13 47.1570 94.3140 141.471 188.628	
5.	Below you can see the figure which includes our command file with all of our concepts/parameters/data.	
6.		

```
DEBUT();

material=DEFI_MATERIAU(ELAS=_F(E=2e05,
                               NU=0.3,),);

mesh=LIRE_MAILLAGE(FORMAT='MED',);

mesh=DEFI_GROUP(reuse =mesh,
                MAILLAGE=mesh,
                CREA_GROUP_MA=_F(NOM='TOUT',
                                 TOUT='OUI',),
                CREA_GROUP_NO=_F(TOUT_GROUP_MA='OUI',),);

model=AFFE_MODELE(MAILLAGE=mesh,
                  AFFE=_F(TOUT='OUI',
                          PHENOMENE='MECANIQUE',
                          MODELISATION='POU_D_T',),);

mater=AFFE_MATERIAU(MAILLAGE=mesh,
                    AFFE=_F(TOUT='OUI',
                            MATER=material,),);

Guide=AFFE_CARA_ELEM(MODELE=model,
                     POUTRE=_F(GROUP_MA='TOUT',
                               SECTION='CERCLE',
                               CARA='R',
                               VALE=15,),);

Fixed=AFFE_CHAR_MECA(MODELE=model,
                     DDL_IMPO=_F(GROUP_NO='Fix',
                                 DX=0,
                                 DY=0,
                                 DZ=0,
                                 DRX=0,
                                 DRY=0,
                                 DRZ=0,),);

Load=AFFE_CHAR_MECA(MODELE=model,
                    FORCE_NODALE=_F(GROUP_NO='Load',
                                    FY=-500,),);

result=MECA_STATIQUE(MODELE=model,
                     CHAM_MATER=mater,
                     CARA_ELEM=Guide,
                     EXCIT=(_F(CHARGE=Fixed,),
                            _F(CHARGE=Load,),),);

result=CALC_CHAMP(reuse =result,
                  RESULTAT=result,
                  CONTRAINTE='SIPO_NOEU',
                  CRITERES='SIEQ_NOEU',
                  FORCE='REAC_NODA',);

IMPR_RESU(FORMAT='MED',
          RESU=_F(MAILLAGE=mesh,
                  RESULTAT=result,
                  NOM_CHAM=('DEPL','REAC_NODA','SIEQ_NOEU','SIPO_NOEU',),),);

FIN();
#CHECKSUM:1250ebc8acca449d92118826032ee9c2  -:FIN CHECKSUM
```

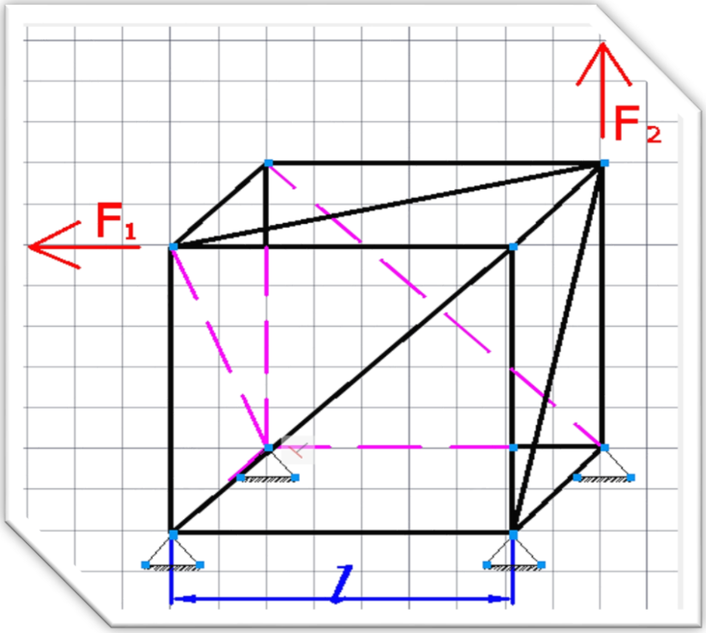
Notes

i. Code_Aster can read IDEAS, GMSH, MED Formats.
ii. If in Eficas you are having trouble understanding a word, use translation from French to English, it is really helpful.
iii. In defining material step, we filled only 2 parameters (Young's Modulus and Poison's Ration) because the task didn't require extra ones like Density, Thermal Conductivity, etc.
iv. There will be cases where you will have two different material types. To do so you will just create another DEFI_MATERIAU concept with different parameters and you will just give it another name.
v. This can happen to other commands also, like in AFFE_MODELE. But have in mind that you can separate groups and give different parameters to each one of them only by changing their names.
vi. The reason why we define mesh group is that we can convert it to node group later.
vii. We need to be careful of which type of modelization we choose, because in most of the cases we pick the wrong one and the command file is not working. To choose the best one in a certain case study, you will have to study carefully the Code_Aster documentation, U4.41.01.
viii. Beams and Shells have 6 Degrees of Freedom, 3 in Translation and another 3 in Rotation.
ix. In Aster, regarding the default 128MB memory parameter, you could leave it like this but soon, for bigger analysis, that amount of memory will be insufficient and it will stop running. The more memory you input the safer it is. Have in mind that the most sufficient amount of memory is around quarter or half of the total memory of your system.
x. There will be times that your analysis will not allow you to visualize Post-Pro Module, that is happening because there is an error in your command file.
xi. When you Run an analysis, it does not matter if it is working or not, Salome_Meca creates some files in the folders directory. [.hdf, .rmed, .resu, .mess, .mmed, .export, .comm file].
xii. In the .mess file as we mentioned before, there will be information regarding your analysis. At the end of the file there will be a "box", in which will be stated the area, the line where your code is mistaken. In the next chapters you will see examples of that file.

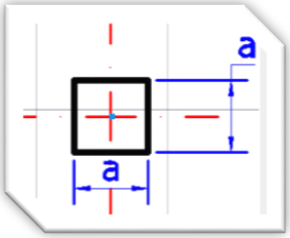
Chapter_IV: Beam System



In this Chapter we will deepen more into beams.
We will study a 1D beam system analysis, emphasizing in Nominal Stress and Alignment.



Square Cross-Section



Step 1: Purpose of the FE Analysis/ Description of the problem

In this chapter we are asked to create a 3D Cube Beam System. The Cube Beam System has a square Cross-Section which is given above.

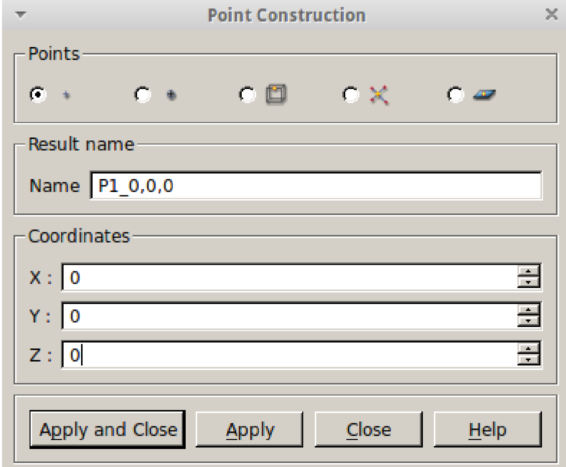
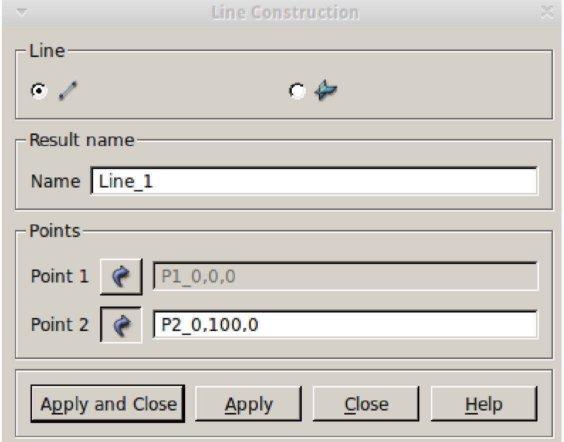
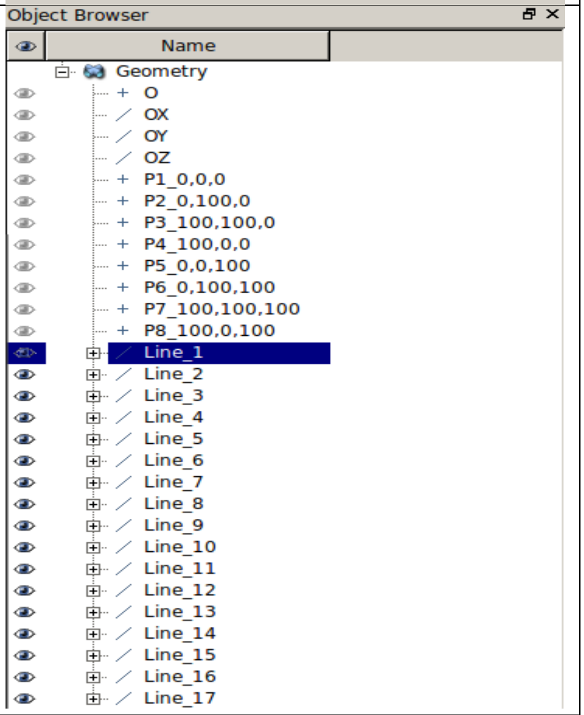
Step 2: Input Values for the FE Analysis

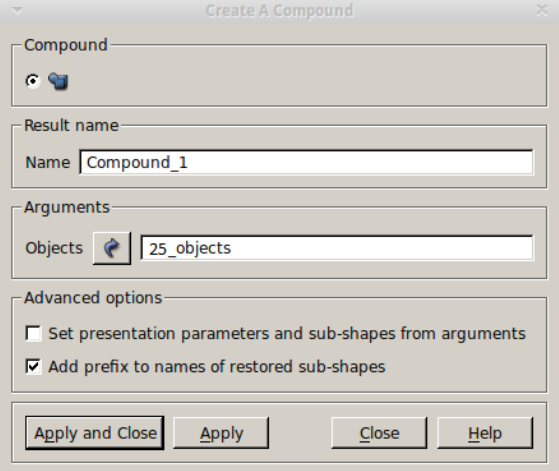
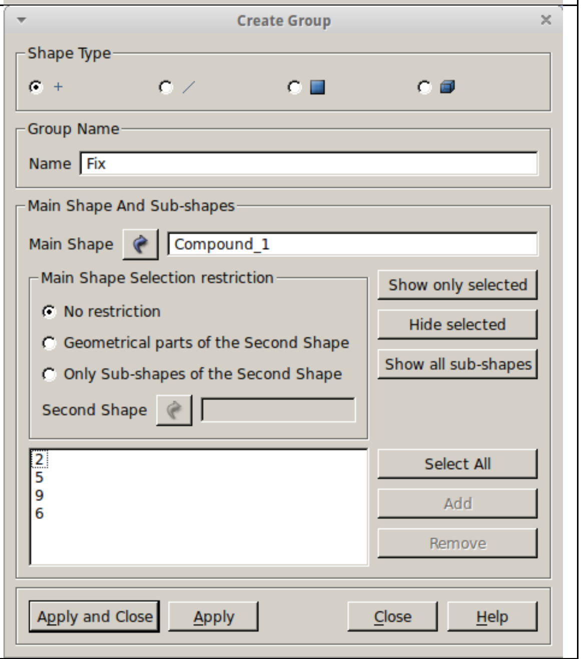
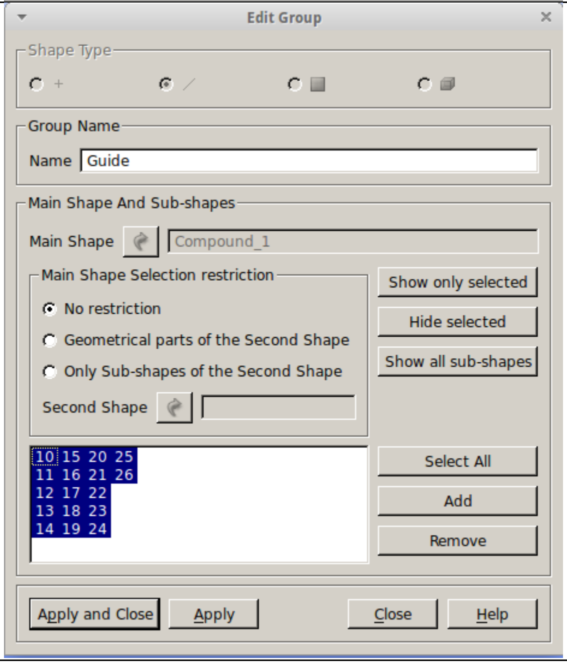
Cube Dimension:	100mm x 100mm x 100mm;
Cross-Section:	Rectangular, Length= 8mm x 8mm;
Young's Modulus:	2e05 MPa;
Poisson's Ratio:	0.3;
Pressure Applied:	F1= 150 N; F2= 200 N;

Two different load cases.

Step 3: Model Geometry

- # As we proceed forward and you learn more and more things, the explanation of the steps will be shorter.
- # Start Salome_Meca and begin modelling.

#	Description	Figure
1.	Starting with, we must add all cube parts in Geometry Module. Start Geometry Module, click on New Entity → Basic → Point. Enter the following Points: **Name** = *Coordinates* (x, y, z) **P1_0,0,0** = *(0,0,0)* **P2_0,100,0** = *(0,100,0)* **P3_100,100,0** = *(100,100,0)* **P4_100,0,0** = *(100,0,0)* **P5_0,0,100** = *(0,0,100)* **P6_0,100,100** = *(0,100,100)* **P7_100,100,100** = *(100,100,100)* **P8_100,0,100** = *(100,0,100)* *Finish with "Apply and Close"*	 Point Construction Points Result name Coordinates Apply and Close Apply Close Help Name: P1_0,0,0 X : 0 Y : 0 Z : 0
2.	After creating the Points we must make Lines out of those Points. Go to New Entity → Basic → Lines Connect all Points according to the figure on the right and our given data. **Line_1** = *(P1,P2)* **Line_2** = *(P2,P3)* **Line_3** = *(P3,P4)* **Line_4** = *(P4,P1)* **Line_5** = *(P1,P5)* **Line_6** = *(P5,P6)* **Line_7** = *(P6,P7)* **Line_8** = *(P7,P8)* **Line_9** = *(P8,P5)* **Line_10** = *(P2,P6)* **Line_11** = *(P4,P8)* **Line_12** = *(P3,P7)* **Line_13** = (P5,*P7)* **Line_14** = *(P8,P3)* **Line_15** = *(P4,P2)* **Line_16** = *(P1,P6)* **Line_17** = *(P6,P3)*	 Line Construction Line Result name Points Apply and Close Apply Close Help Name: Line_1 Point 1: P1_0,0,0 Point 2: P2_0,100,0
3.	After finishing with the Points, Lines you should be able to see the geometry as shown in the figure on the right.	 Geometry O OX OY OZ P1_0,0,0 P2_0,100,0 P3_100,100,0 P4_100,0,0 P5_0,0,100 P6_0,100,100 P7_100,100,100 P8_100,0,100 Line_1 Line_2 Line_3 Line_4 Line_5 Line_6 Line_7 Line_8 Line_9 Line_10 Line_11 Line_12 Line_13 Line_14 Line_15 Line_16 Line_17

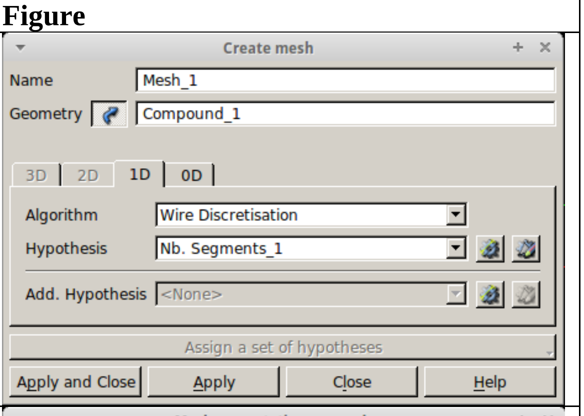
#	Description	Figure
4.	Next we need to create a container that will store all these individual lines. Click New Entity → Build → Compound. Keep default name. Select all our Lines and Points for Objects. *Finish with "Apply and Close"*	 Create A Compound Compound ● Result name Name: Compound_1 Arguments Objects: 25_objects Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes Apply and Close · Apply · Close · Help
5.	Next we need to create Groups of Nodes for our Geometric and Load Boundary Conditions. Click New Entity → Groups → Create. In the window that pops-up, select the 1st algorithm for Shape Type of Nodes. Enter "*Fix*" as a name (or any name that suits you) and select the Points where we want our fixed support to be placed, from the Object Browser. Then click on "*Add*" and the Points will be added likewise the figure on the right. *Click "Apply"* Similarly create Node group of our loads, for Load_F1 and Load_F2. *Click "Apply"*	 Create Group Shape Type ● ○ ○ ○ Group Name Name: Fix Main Shape And Sub-shapes Main Shape: Compound_1 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape Show only selected · Hide selected · Show all sub-shapes 2 5 9 6 Select All · Add · Remove Apply and Close · Apply · Close · Help
6.	Select now the 2nd Algorithm for Shape Type to select Edges. Name it "*Guide*" (or any name that suits you), click "Select All" and then on "*Add*". All Lines/Edges will be added now on the list like the figure on the right. *Finish with "Apply and Close"*	 Edit Group Shape Type ○ ● ○ ○ Group Name Name: Guide Main Shape And Sub-shapes Main Shape: Compound_1 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape Show only selected · Hide selected · Show all sub-shapes 10 15 20 25 11 16 21 26 12 17 22 13 18 23 14 19 24 Select All · Add · Remove Apply and Close · Apply · Close · Help
7.	This concludes our Geometry creation session. Save in a proper location with a proper name.	

Step 4: Meshing Geometry

<table>
<tr><th>#</th><th>Description</th><th>Figure</th></tr>
<tr><td><i>1.</i></td><td>Select and open Mesh Module from the drop down list. Click Mesh → Create Mesh (with Compound_1 selected).

Select “<i>Wire Discretisation</i>” for 1D Algorithm.

Select Nb. Segments_1 with number of Segments “<i>10</i>” as Hypothesis.

<i>Finish with “Apply and Close”</i></td><td>

Name: Mesh_1
Geometry: Compound_1

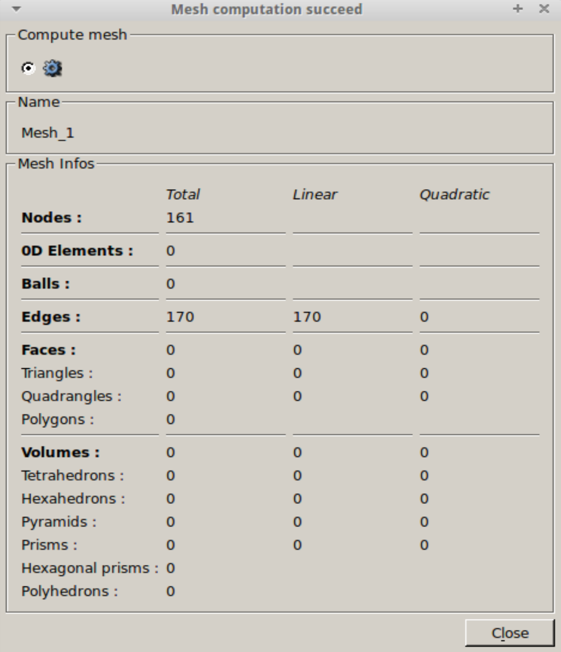
Algorithm: Wire Discretisation
Hypothesis: Nb. Segments_1
Add. Hypothesis: <None>

Assign a set of hypotheses
Apply and Close Apply Close Help

</td></tr>
<tr><td><i>2.</i></td><td>Right-click on Mesh_1 → Compute.

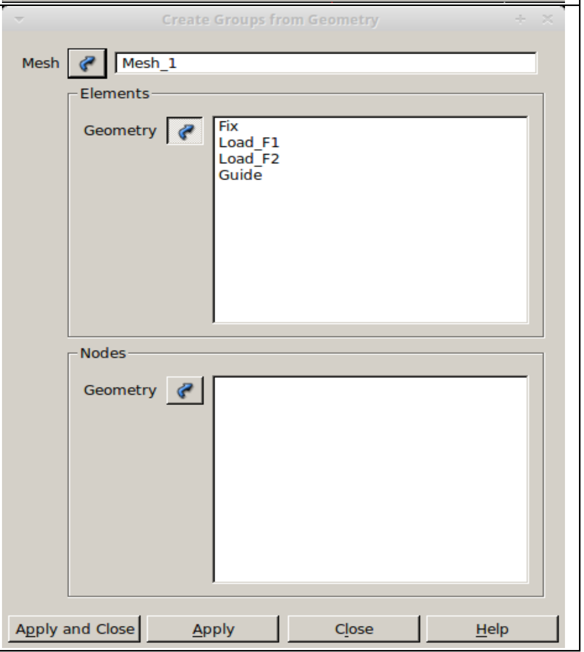
If everything is correct you will have the same results as in the figure on the right.

We know that we have 17 edges/lines with 10 Nb.Segments, so there should be 170 total edges.

<i>Click “Close”</i></td><td>

Name: Mesh_1
Mesh Infos
<table>
<tr><th></th><th><i>Total</i></th><th><i>Linear</i></th><th><i>Quadratic</i></th></tr>
<tr><td>Nodes :</td><td>161</td><td></td><td></td></tr>
<tr><td>0D Elements :</td><td>0</td><td></td><td></td></tr>
<tr><td>Balls :</td><td>0</td><td></td><td></td></tr>
<tr><td>Edges :</td><td>170</td><td>170</td><td>0</td></tr>
<tr><td>Faces :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Triangles :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Quadrangles :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Polygons :</td><td>0</td><td></td><td></td></tr>
<tr><td>Volumes :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Tetrahedrons :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Hexahedrons :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Pyramids :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Prisms :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Hexagonal prisms :</td><td>0</td><td></td><td></td></tr>
<tr><td>Polyhedrons :</td><td>0</td><td></td><td></td></tr>
</table>
</td></tr>
<tr><td><i>3.</i></td><td>Right-click on Mesh_1 → Create Groups From Geometry.

Select “<i>Fix</i>”, “<i>Load_F1</i>”, “<i>Load_F2</i>”, “<i>Guide</i>” from Geometry’s Object Browser and the groups will be added likewise the figure on the right.

<i>Finish with “Apply and Close”</i></td><td>

Mesh: Mesh_1
Elements
Geometry: Fix
Load_F1
Load_F2
Guide
Nodes
Geometry:
 </td></tr>
</table>

Step 5,6,7,8 and 9: Creating the command (.comm) file

- Up until now we have seen a couple of ways to create the command file. At the beginning we used Linear Elastic Wizard from Salome_Meca, later we used the help of Eficas Application. Now we will edit by hand in a text editor an existing command file (the one from the previous chapter), and we will modify it to fit our new demands.
- First thing you need to do is Copy the command file from the previous chapter and Paste it to the folder of this case.
- Rename it with an appropriate name and open it with a text editor.
- Red letters are the new modifications and
- Are commentary to help you understand what we are doing.

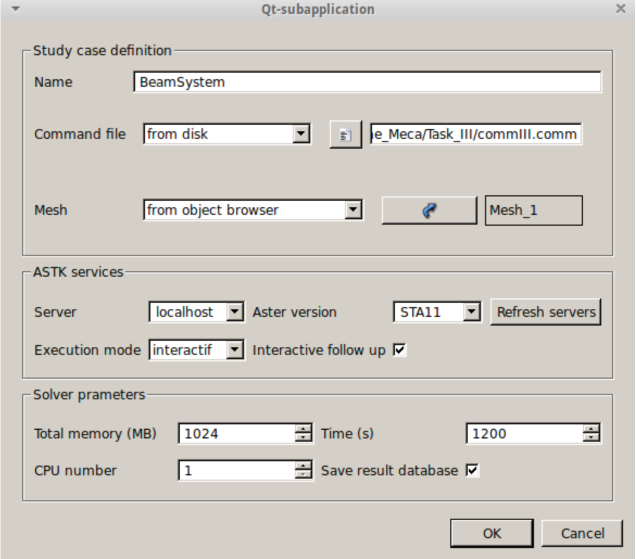
```
DEBUT ();

material = DEFI_MATERIAU ( ELAS= _F ( E = 2e05,
                                      NU = 0.3, ), );
mesh = LIRE_MAILLAGE ( FORMAT = 'MED', );

mesh = DEFI_GROUP ( reuse = mesh,
                    MAILLAGE = mesh,
                    CREA_GROUP_MA = _F ( NOM = 'TOUT',
                                         TOUT = 'OUI', ),
                  CREA_GROUP_NO= _F ( TOUT_GROUP_MA = 'OUI', ), );

model = AFFE_MODELE ( MAILLAGE = mesh,
                      AFFE = _F ( TOUT = 'OUI',
                                  PHENOMENE = 'MECHANIQUE',
                                  MODELISATION = 'POU_D_T', ), );

mater = AFFE_MATERIAU ( MAILLAGE = mesh,
                        AFFE = _F ( TOUT = 'OUI',
                                    MATER = material, ), );
#We now want rectangular Cross-Section, so we need to change CERCLE --> RECTANGULAR and define our new
dimensions.

Guide = AFFE_CARA_ELEM ( MODEL = model,
                         POUTRE = _F ( GROUP_MA = 'TOUT',
                                       SECTION = 'RECTANGLE',
                                       CARA = ( 'HY', 'HZ', ),
                                       VALE = ( 8, 8, ), ), );          # VALE = Values

Fixed = AFFE_CHAR_MECA ( MODEL = model,
                         DDL_IMPO = _F ( GROUP_NO = 'Fix',
                                         DX = 0,
                                         DY = 0,
                                         DZ = 0,
                                         DRX = 0,
                                         DRY = 0,
                                         DRZ = 0, ), );

```

```
#In this case study we have different load cases. We got 2 different forces now F1 and F2 (Load_F1/F2).
#Keep FORCE_NODALE concept but edit the parameters. Load_F1 in -X direction and Load_F2 in Y direction.
#Magnitude F1=150 N, F2=200 N.

Load = AFFE_CHAR_MECA ( MODEL = model,
                               FORCE_NODALE = ( _F ( GROUP_NO = 'Load_F1',
                                                     FX = -150, ),
                                                 _F ( GROUP_NO = 'Load_F2',
                                                     FY = 200, ), ), );

result = MECA_STATIQUE ( MODEL = model,
                            CHAM_MATER = mater,
                            CARA_ELEM = Guide,
                            EXCIT = ( _F ( CHARGE = Fixed, ),
                                      _F ( CHARGE = Load, ), ), );

result = CALC_CHAMP ( reuse = result,
                         RESULTAT = result,
                         CONTRAINTE = ’SIPO_NOEU’,
                         FORCE = ’REAC_NODA’, );

IMPR_RESU = ( FORMAT = ’MED’,
                RESU = _F ( MAILLAGE = mesh,
                            RESULTAT = result,
                            NOM_CHAM = ( ‘DEPL’, ’REAC_NODA’, ’SIPO_NOEU’, ), ), );

FIN ();
```

# Save this file and run the FE Analysis now that we have input all the required data.

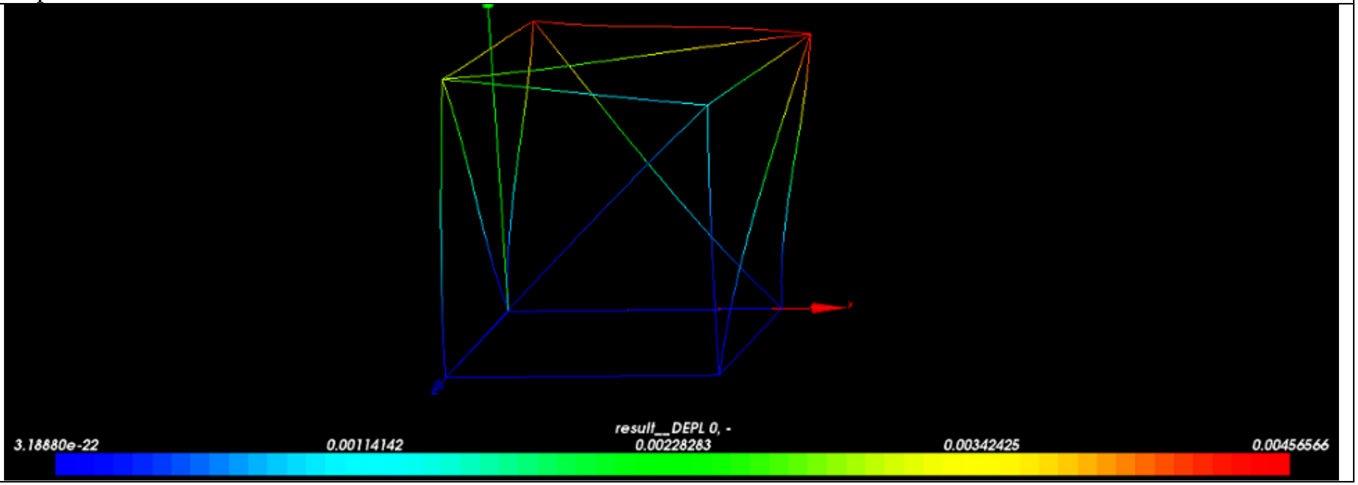
Step 10: Run the Analysis

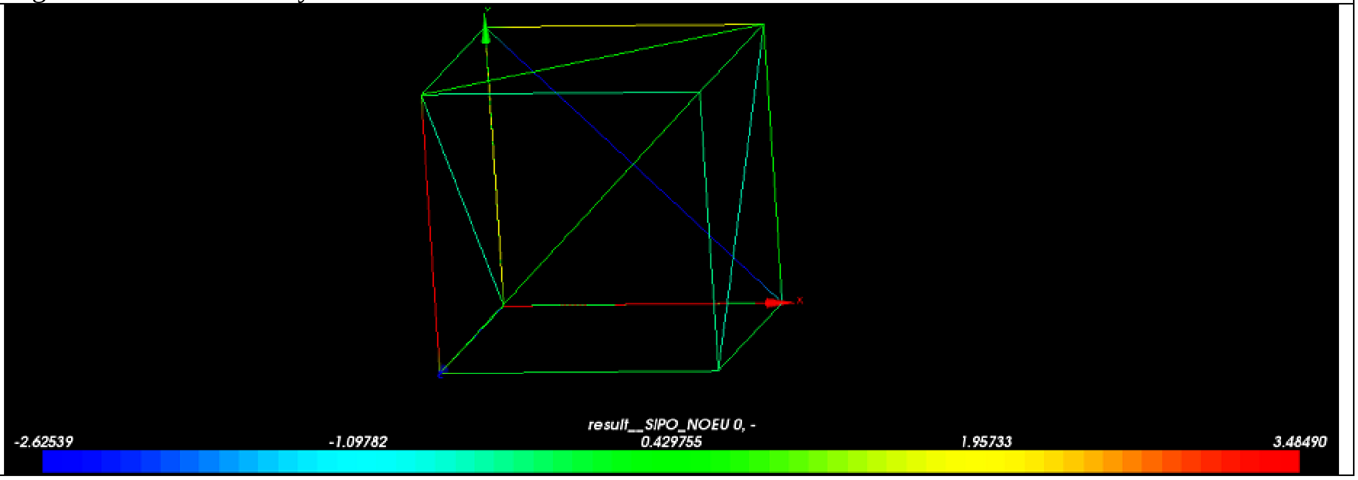
# At this point in order to run the FE Analysis we need a Mesh file, and a command file. Since we have both of those files let's start the analysis and then proceed to the visualization mode.

#	Description	Figure
1.	Activate Aster Module, go to Aster → Add Study Case Fill in the gaps according to the figure on the right. Give it a considerate name, choose the command file from its folder location, choose the mesh file from the Object Browser. Change to STA11 of Aster Version, and lastly increase the Total Memory (MB) and Time (s). *Finish with "Ok"*	 Study case definition Name BeamSystem Command file from disk e_Meca/Task_III/commIII.comm Mesh from object browser Mesh_1 ASTK services Server localhost Aster version STA11 Refresh servers Execution mode interactif Interactive follow up ☑ Solver prameters Total memory (MB) 1024 Time (s) 1200 CPU number 1 Save result database ☑
2.	A new branch named "*BeamSystem*" (the choosen name) will be created in the Object Browser. Right-click on it → Run, to start the Analysis. When the analysis is done and everything has been done correctly a new branch named "*Post-Pro*" may appear.	

Step 11: Post Processing the results

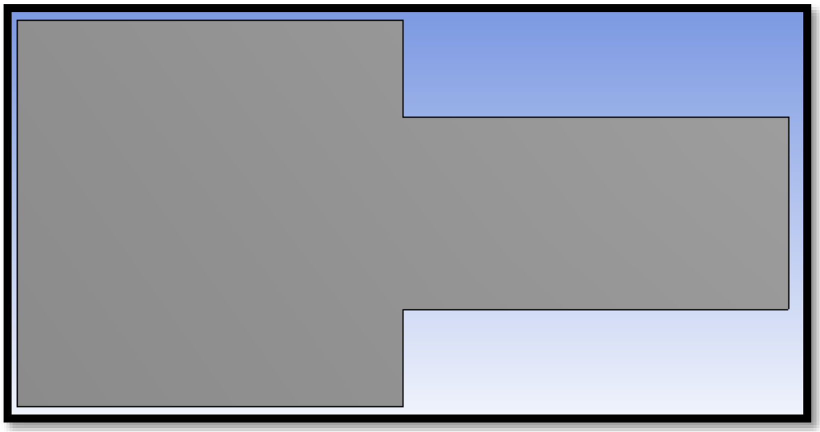
# Now that the FE Analysis is complete let's have a look at the visualized results Post-Pro Module is offering us.

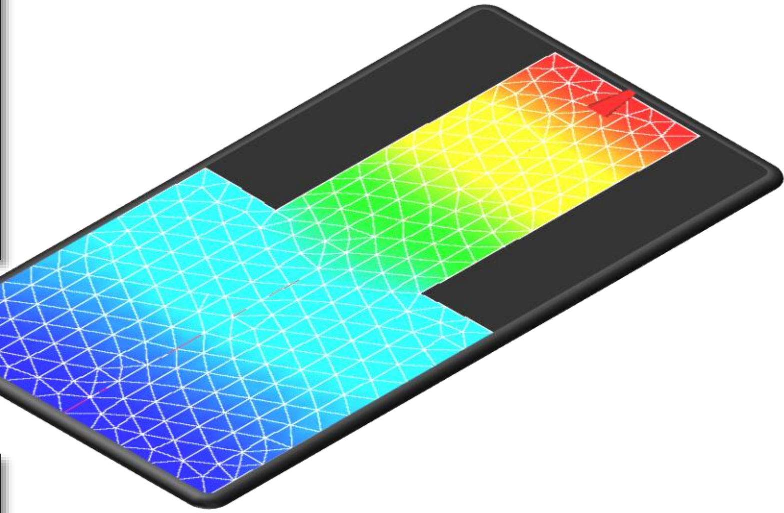
#	Description	Figure
1.	We will not do anything new in this step, that we haven't already done. Activate Post-Pro Module from the drop down list, open result__DEPL branch, right-click on 0,- → Deformed Shape → tick Magnitude coloring → "Apply and Close" → right-click on Defor.Shape → Show Only → Representation → Surface.	
2.	 result__DEPL 0, - 3.18880e-22 0.00114142 0.00228283 0.00342425 0.00456566	

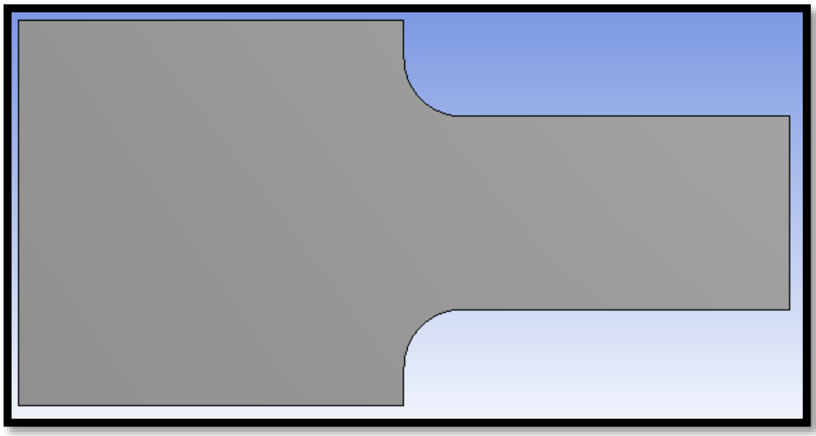
#	Description	Figure
3.	Same process for the Stresses also. Open result__SIPO_NOEU → Right-click on 0,- → Scalar Map. In this case we will use "*SN*" as Scalar Mode. Right-click → Show Only.	
4.	 result__SIPO_NOEU 0, - -2.62539 -1.09782 0.429755 1.95733 3.48490	

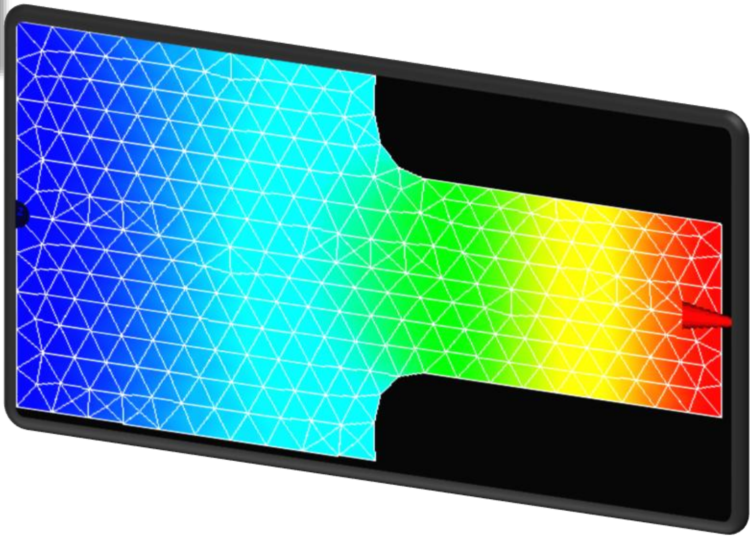
Chapter_V: Plate

In this Chapter we are interested in a Plane Stress 2D Analysis, with a notch in the geometry and Singularity.







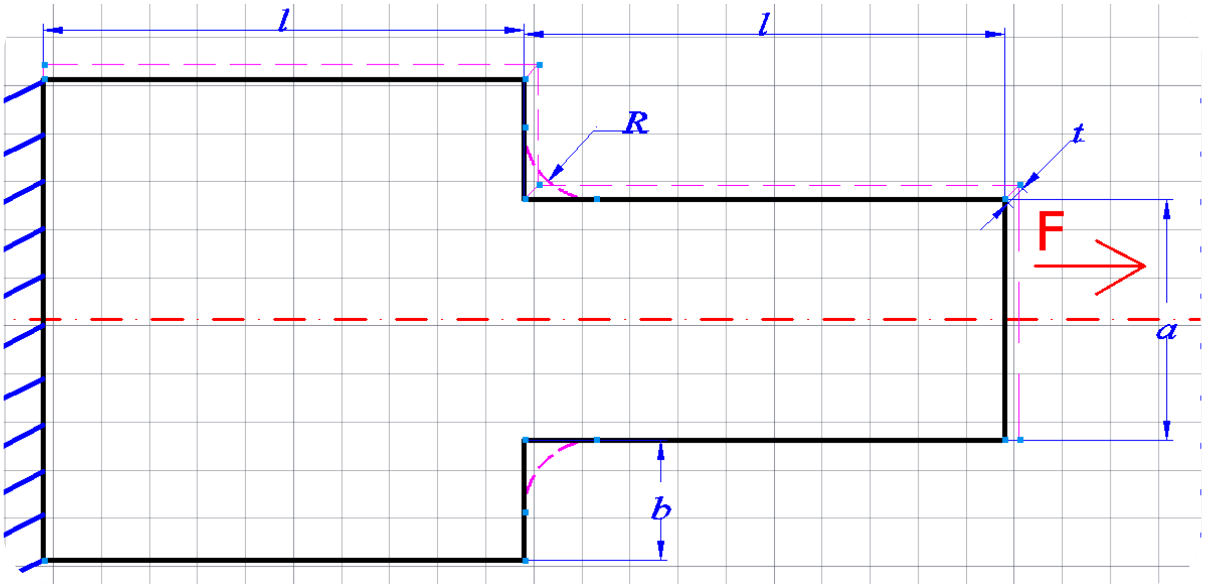


Step 1: Purpose of the FE Analysis/ Description of the problem

In this chapter we are interested to know the concepts of Finite Element Simulation, like “Stress Discontinuity”, “Structural Error”, “Stress Singularity”, “Finite Element Convergence”. To demonstrate this concepts we are going to use a Plate geometry with notch, which is made out of steel and has the given dimensions. The bar has Fixed Support on the left side and is subject to a tension of 50MPa on the right side.

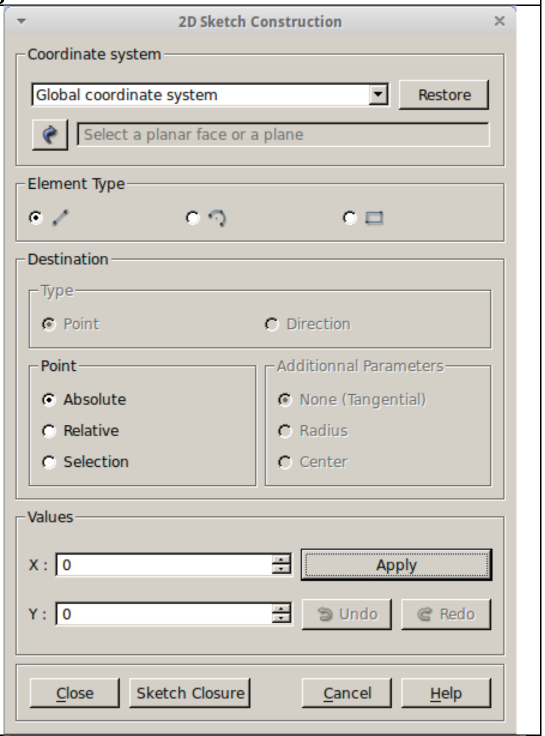
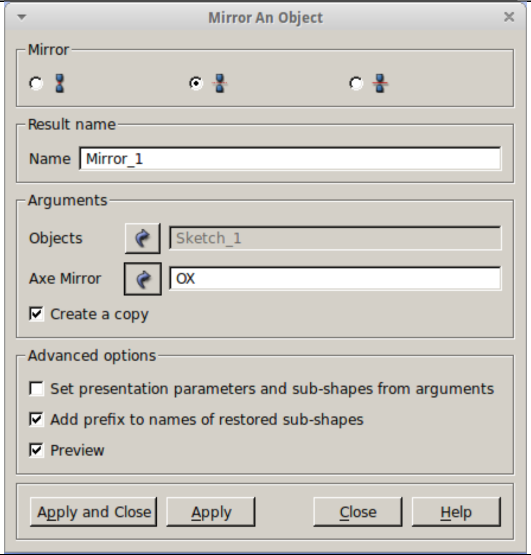
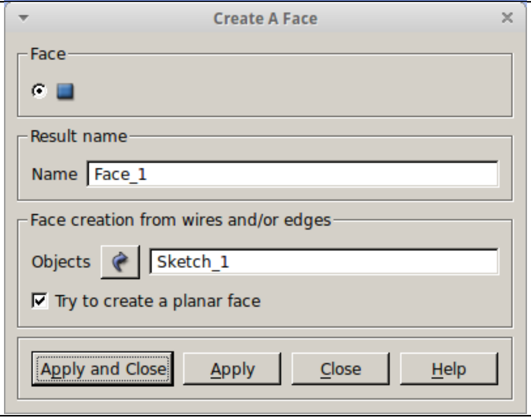
Step 2: Input Values for the FE Analysis

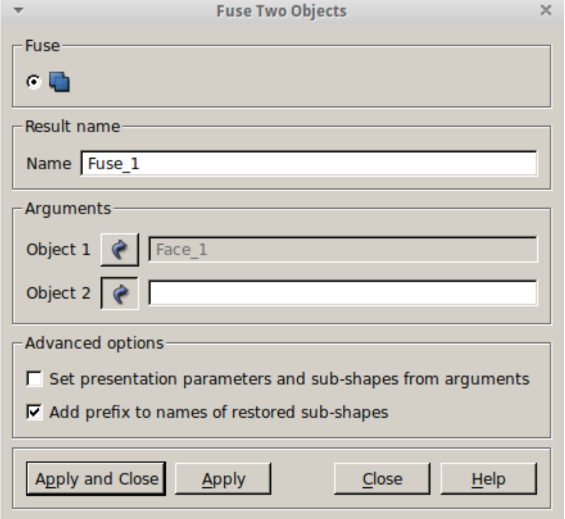
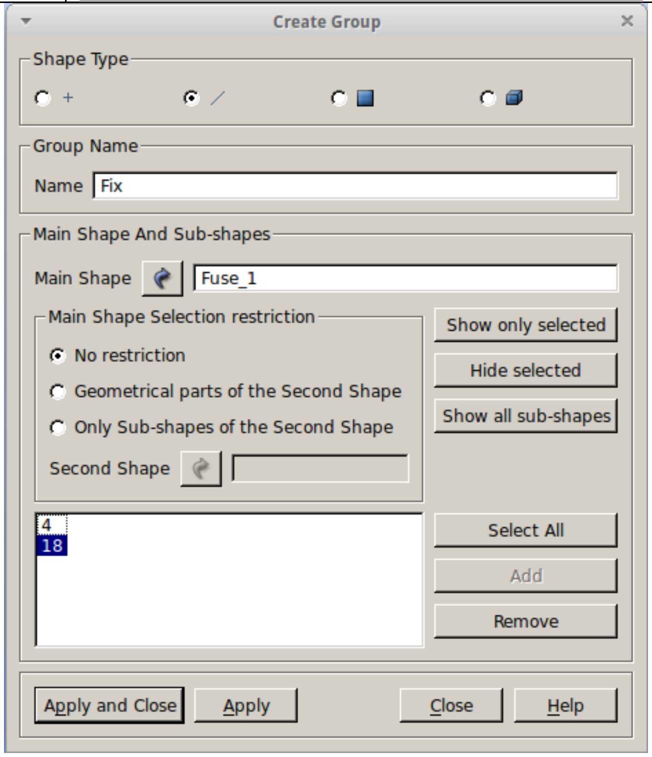
Dimension:	Length = 100 mm [l]; Length = 50 mm [a]; Radius = 15 mm [R];	Length = 25 mm [b]; Thickness = 10 mm [t];
Young’s Modulus:	200 GPa;	
Poisson’s Ratio:	0.3;	
Force Applied:	50 N [F];	



Step 3: Model Geometry

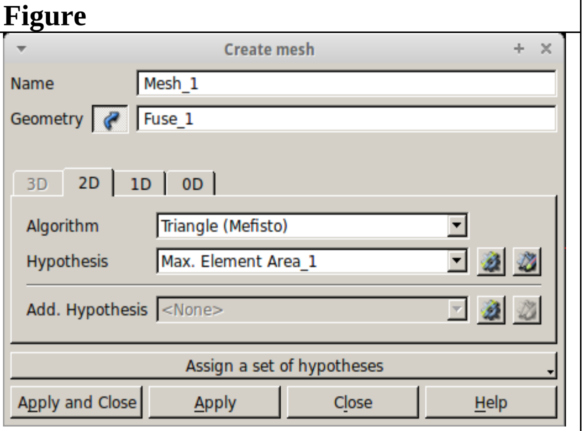
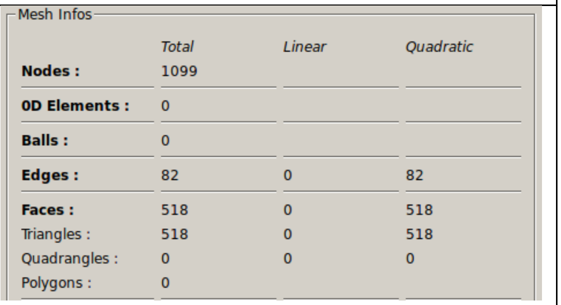
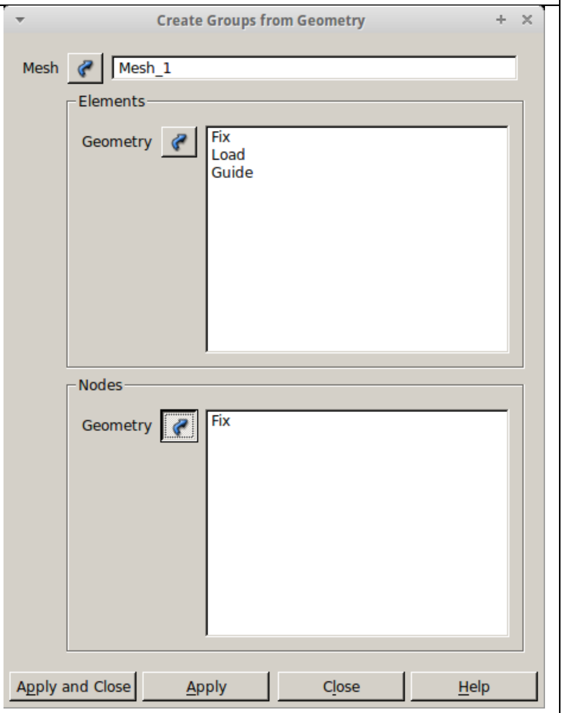
# Open up Salome_Meca and start the input for Bar Shell Model.

#	Description	Figure
1.	To generate a Shell model you just need to create a planar face. Click New Entity → Basic → 2D_Sketch. In the window that pops-up keep “Global coordinate system”, “Element Type” and “Point” default. Enter for first value, **P_1 → X=0 _ Y=0** *Click “Apply”* Your next points should be: **P_2 → X=0 _ Y=50** **P_3 → X=100 _ Y=50** **P_4 → X=100 _ Y=25** **P_5 → X=200 _ Y=25** **P_6 → X=200 _ Y=0** *Click “Sketch Closure”*	 2D Sketch Construction Coordinate system Global coordinate system Restore Select a planar face or a plane Element Type Destination Type ● Point ○ Direction Point ● Absolute ○ Relative ○ Selection Additionnal Parameters ● None (Tangential) ○ Radius ○ Center Values X : 0 Y : 0 Apply Undo Redo Close Sketch Closure Cancel Help
2.	We need to create a mirror image. Go to Operations → Transformation → Mirror Image. Select 2nd Algorithm for Mirror. Objects: Sketch_1. Axe Mirror: OX Plane. Select “Create a copy”. *Click “Apply and Close”*	 Mirror An Object Mirror Result name Name Mirror_1 Arguments Objects Sketch_1 Axe Mirror OX ☑ Create a copy Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes ☑ Preview Apply and Close Apply Close Help
3.	Next you need to convert this Wires into a Face. Go to New Entity → Build → Face. Keep default name, and in the Object Browser, select Sketch_1. *Click “Apply”* Do the same for Face_2 with Mirror_1 for “Objects”. Select “Try to create a planar face”. *Click “Apply and Close”*	 Create A Face Face Result name Name Face_1 Face creation from wires and/or edges Objects Sketch_1 ☑ Try to create a planar face Apply and Close Apply Close Help

#	Description	Figure
4.	Next part is to create a shape out of the two faces we assigned. Operations → Boolean → Fuse will do the trick. Leave default name, and assign the Arguments accordingly. Face_1 for "Object 1". Face_2 for "Object 2". *Finish with "Apply and Close"*	Fuse Two Objects Result name Name: Fuse_1 Arguments Object 1: Face_1 Object 2: Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes
5.	Next part will be, creating our Geometry groups, which will help us to assign supports, load cases, forces, etc. Go to New Entity → Group → Create. In the Create Group window that pops-up, select 2nd Algorithm for lines. Enter Fix as Group Name, and then select and add each one of the two left lines where we want our Fix Support to be. We got two lines here and not only one because we used Mirror Image to create our Geometry. Click "Apply". Likewise select the opposite lines and name them Load. Change the Shape Type algorithm to the 3rd option, the Face selection. Name your group Guide and Add the two face available to you. *Finish with "Apply and Close"*	Create Group Shape Type: ○ + ● ⁄ ○ ○ Group Name Name: Fix Main Shape And Sub-shapes Main Shape: Fuse_1 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape: 4 18 Show only selected Hide selected Show all sub-shapes Select All Add Remove Apply and Close Apply Close Help

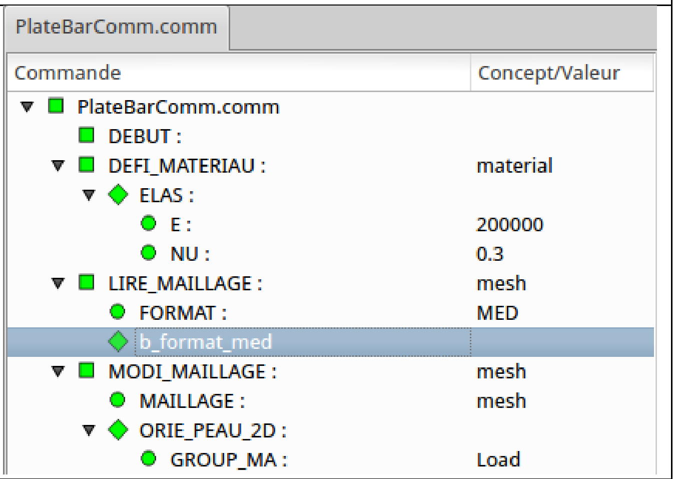
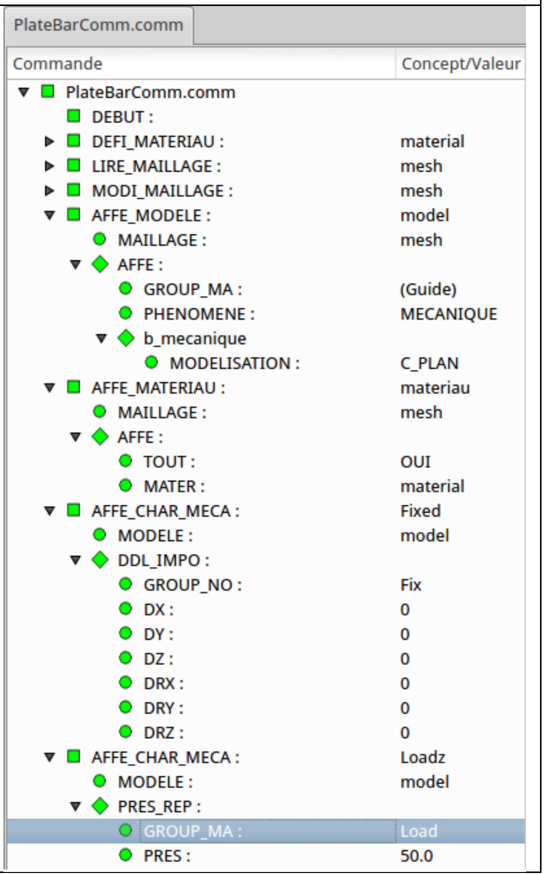
Step 4: Meshing the Geometry

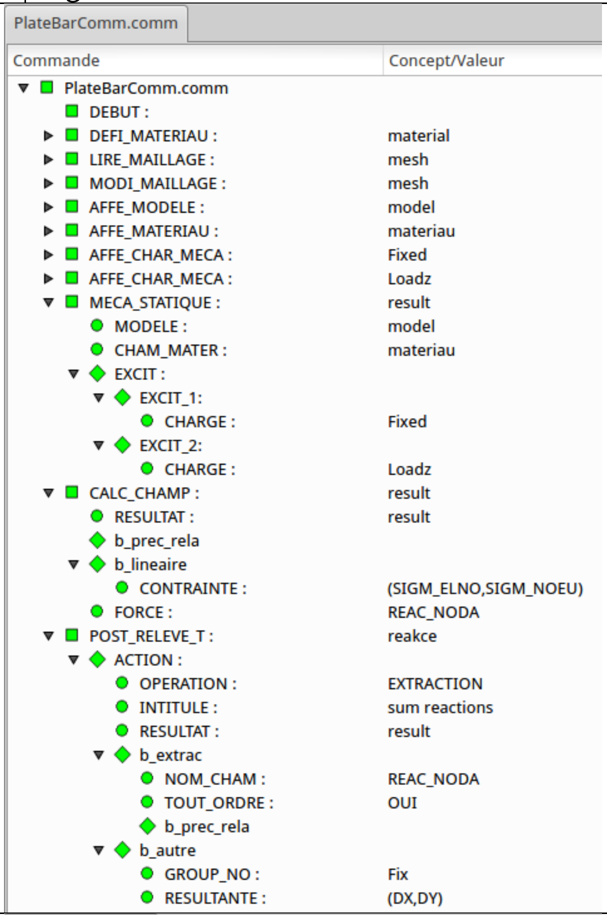
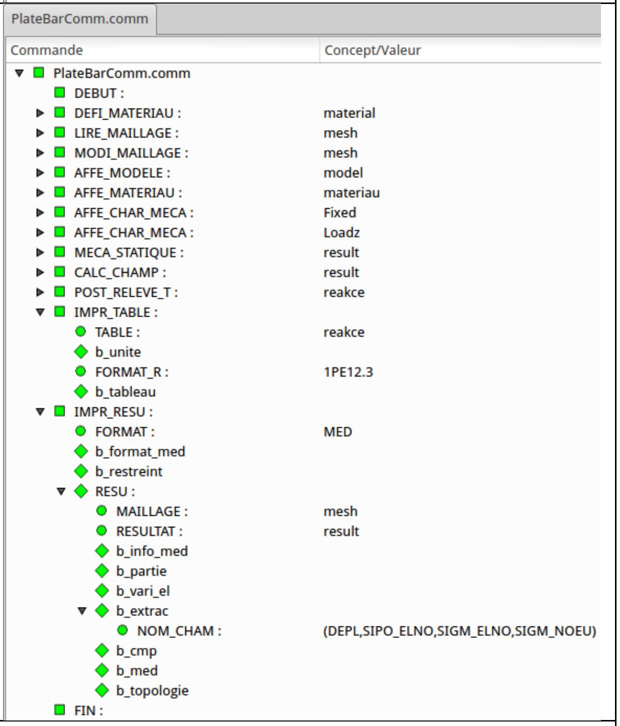
# In this step, we are going to use all of our resources to set up the mesh and then create our command file (.comm). Let's get started by opening Mesh module from the drop down list.

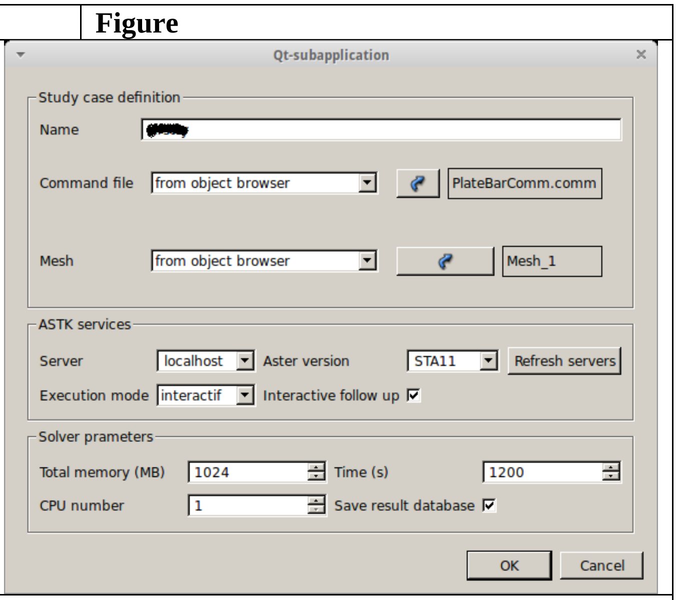
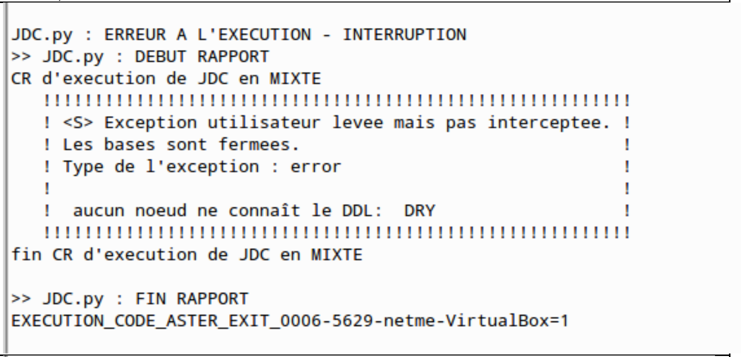
#	Description	Figure
1.	Go to Mesh → Create Mesh. Fill in the gaps like the figure on the right. But this time when it is time to input Max. Element Area_1, change the default value of ***Max. Area=20***. For 1D tab, choose Wire Discretisation for Algorithm and Hypothesis would be **Max. Size=10**. *Click "Ok"* *Finish with "Apply and Close"*	Create mesh Name: Mesh_1 Geometry: Fuse_1 3D 2D 1D 0D Algorithm: Triangle (Mefisto) Hypothesis: Max. Element Area_1 Add. Hypothesis: <None> Assign a set of hypotheses Apply and Close / Apply / Close / Help
2.	Now right-click on Mesh_1 → Compute. Mesh information window will pop-up. Right-click on Mesh_1 and select "Convert from/to quadratic". *Click "Apply and Close"* Compute Mesh_1 again, and you will have results according to the figure on the right.	Mesh Infos Total / Linear / Quadratic **Nodes :** 1099 **0D Elements :** 0 **Balls :** 0 **Edges :** 82 / 0 / 82 **Faces :** 518 / 0 / 518 Triangles : 518 / 0 / 518 Quadrangles : 0 / 0 / 0 Polygons : 0
3.	Last thing we need to do, is to create groups from our Geometry. Right-click on Mesh_1 → Create Groups From Geometry. Select "Fix", "Load", "Guide" from the Object Browser and add them to the window as shown in the figure on the right. Unselect the blue curved arrow next to "Geometry", to be able to select Fix for the Nodes. *Finish with "Apply and Close"*	Create Groups from Geometry Mesh: Mesh_1 Elements — Geometry: Fix, Load, Guide Nodes — Geometry: Fix Apply and Close / Apply / Close / Help
4.	Save your work, and continue to the next step.	

Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis

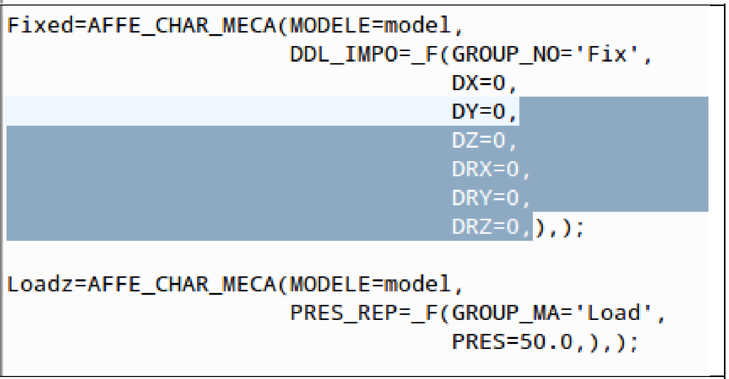
# Although we gained some experience in creating the command file with a couple of different tools, opening a text editor and start coding, is still a long way down the road. Eficas Application will do the trick again, and we will also use a text editor to edit some points, to get more familiar with coding.

#	Description	Figure
1.	Activate Eficas module from the drop down list. Go to Eficas → Eficas MultiCatalogue. In the pop-up window, click on Create New or Nouveau. Choose ASTER → "Ok", STA11 → "Ok" and you have set up Eficas and it is ready for modification. Before starting editing, save the command file in the same location where the geometry/mesh/.hdf files are currently located, with an appropriate name. Fishier → Enregistrer sous.	
2.	The figure on the right shows the first commands that we need to conclude in the command file. DEBUT → Begin. DEFI_MATERIAU → Define the material. Structural Steel is the material we will use in this case study. LIRE_MAILLAGE → Read the mesh, with MED Format. MODI_MAILAGE → Modify the mesh. *ORIE_PEAU_2D → Reorientation of mesh, to be able to apply pressure on a "Skin".*	 PlateBarComm.comm Commande — Concept/Valeur PlateBarComm.comm DEBUT : DEFI_MATERIAU : material ELAS : E : 200000 NU : 0.3 LIRE_MAILLAGE : mesh FORMAT : MED b_format_med MODI_MAILLAGE : mesh MAILLAGE : mesh ORIE_PEAU_2D : GROUP_MA : Load
3.	AFFE_MODELE → Apply/ Assign to the model, the mesh, phenomenon and type of modelization. In our case we are dealing with 2D plane elements that is why we use C_PLAN. Also we want Mechanical phenomenon. AFFE_MATERIAU → Apply/ Assign material. AFFE_CHAR_MECA → Apply/ Assign Characteristics, starting with Degrees of Freedom. In this case we want Fixed Support on the "Fix" group. AFFE_CHAR_MECA → Apply/ Assign Characteristics, Load cases. Pressure of 50 MPa to group "Load".	 PlateBarComm.comm Commande — Concept/Valeur PlateBarComm.comm DEBUT : DEFI_MATERIAU : material LIRE_MAILLAGE : mesh MODI_MAILLAGE : mesh AFFE_MODELE : model MAILLAGE : mesh AFFE : GROUP_MA : (Guide) PHENOMENE : MECANIQUE b_mecanique MODELISATION : C_PLAN AFFE_MATERIAU : materiau MAILLAGE : mesh AFFE : TOUT : OUI MATER : material AFFE_CHAR_MECA : Fixed MODELE : model DDL_IMPO : GROUP_NO : Fix DX : 0 DY : 0 DZ : 0 DRX : 0 DRY : 0 DRZ : 0 AFFE_CHAR_MECA : Loadz MODELE : model PRES_REP : GROUP_MA : Load PRES : 50.0

#	Description	Figure
4.	MECA_STATIQUE → Linear static mechanic with superposition on loads. CALC_CHAMP → Create a result by calculating fields by elements or nodes. The “b_lineaire” branch is the command that allows Code_Aster to calculate specific results. In this case we are interested in Averaged (_NOEU) Normal Stress (SIGM_) and Unaveraged (_ELNO). POST_RELEVE_T → This commands summarizes assigned results and puts them to the .resu file, in a tabular or a conventional format. In this case we are interested in summarizing the reaction forces. The first abstract of code, puts the sum of the individual reaction on the group “Fix”, in a table named “reakce”, with OPERATION=’EXTRACTION’ keyword.	 PlateBarComm.comm Commande Concept/Valeur PlateBarComm.comm DEBUT : DEFI_MATERIAU : material LIRE_MAILLAGE : mesh MODI_MAILLAGE : mesh AFFE_MODELE : model AFFE_MATERIAU : materiau AFFE_CHAR_MECA : Fixed AFFE_CHAR_MECA : Loadz MECA_STATIQUE : result MODELE : model CHAM_MATER : materiau EXCIT : EXCIT_1: CHARGE : Fixed EXCIT_2: CHARGE : Loadz CALC_CHAMP : result RESULTAT : result b_prec_rela b_lineaire CONTRAINTE : (SIGM_ELNO,SIGM_NOEU) FORCE : REAC_NODA POST_RELEVE_T : reakce ACTION : OPERATION : EXTRACTION INTITULE : sum reactions RESULTAT : result b_extrac NOM_CHAM : REAC_NODA TOUT_ORDRE : OUI b_prec_rela b_autre GROUP_NO : Fix RESULTANTE : (DX,DY)
5.	IMPR_TABLE → Prints the table of summarized reactions. IMPR_RESU → Stores down and prints the final results. The “b_extrac” branch creates the branches in Post-Pro module. FIN → Finish.	 PlateBarComm.comm Commande Concept/Valeur PlateBarComm.comm DEBUT : DEFI_MATERIAU : material LIRE_MAILLAGE : mesh MODI_MAILLAGE : mesh AFFE_MODELE : model AFFE_MATERIAU : materiau AFFE_CHAR_MECA : Fixed AFFE_CHAR_MECA : Loadz MECA_STATIQUE : result CALC_CHAMP : result POST_RELEVE_T : reakce IMPR_TABLE : TABLE : reakce b_unite FORMAT_R : 1PE12.3 b_tableau IMPR_RESU : FORMAT : MED b_format_med b_restreint RESU : MAILLAGE : mesh RESULTAT : result b_info_med b_partie b_vari_el b_extrac NOM_CHAM : (DEPL,SIPO_ELNO,SIGM_ELNO,SIGM_NOEU) b_cmp b_med b_topologie FIN :
6.	Save the command file. In the next parts, we will check if the command file we created is functional, otherwise we are going to locate our mistakes and try to fix them.	

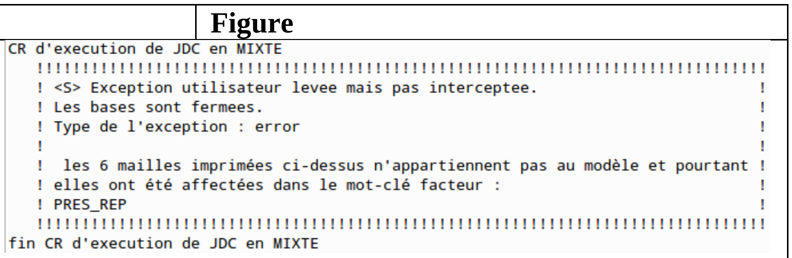
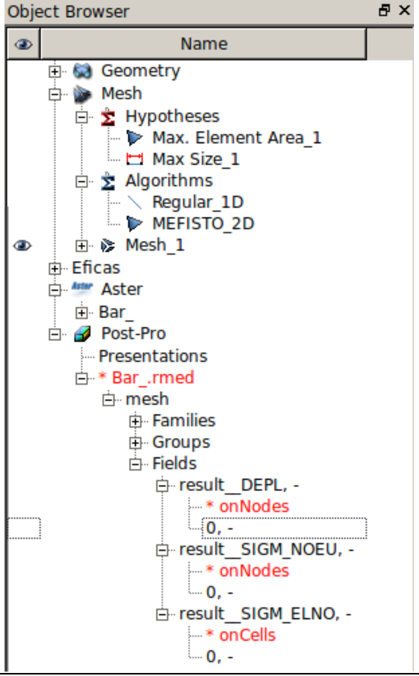
#	Description	Figure
7.	Close Eficas Application and get back to Salome_Meca. Activate Aster Module from the drop down list, go to Aster → Add a Study Case and fill in the gaps according to the figure on your right. *Click "Ok"* A new branch appears in the Object Browser with the same name you indicated before. Right-click on it and select "Run".	Qt-subapplication Study case definition Name [redacted] Command file: from object browser, PlateBarComm.comm Mesh: from object browser, Mesh_1 ASTK services Server: localhost Aster version: STA11 Refresh servers Execution mode: interactif Interactive follow up ☑ Solver prameters Total memory (MB): 1024 Time (s): 1200 CPU number: 1 Save result database ☑ OK Cancel
8.	As you noticed, the analysis is done but Post-Pro branch did not appear like it was supposed to. This is because there is a mistake or let's say a wrong input in our command file that needs fixing. To do so we will have to find a file with the extension .mess, which contains all the analysis information including the errors. This file is located to the same folder with your .comm and .hdf file. After locating the .mess file, open it with a text editor and scroll your way down. At the bottom of the text, you can see a "box" created with exclamation marks. Inside that box lays the information about what went wrong during the analysis. One of the most disturbing issues you will face with Salome_Meca and Code_Aster is that everything is in French, including the analysis information text file (.mess).	
9.	According to my .mess file, there is an error in the DDL_IMP command, regarding DRY orientation. It is fair error if we take into consideration that we are dealing with 2D Plane Elements, meaning that DRY,DRX,DRZ and DZ orientations are redudrant.	

```
JDC.py : ERREUR A L'EXECUTION - INTERRUPTION
>> JDC.py : DEBUT RAPPORT
CR d'execution de JDC en MIXTE
   !!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
   ! <S> Exception utilisateur levee mais pas interceptee. !
   ! Les bases sont fermees.                               !
   ! Type de l'exception : error                           !
   !                                                       !
   !  aucun noeud ne connaît le DDL:  DRY                  !
   !!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
fin CR d'execution de JDC en MIXTE

>> JDC.py : FIN RAPPORT
EXECUTION_CODE_ASTER_EXIT_0006-5629-netme-VirtualBox=1
```

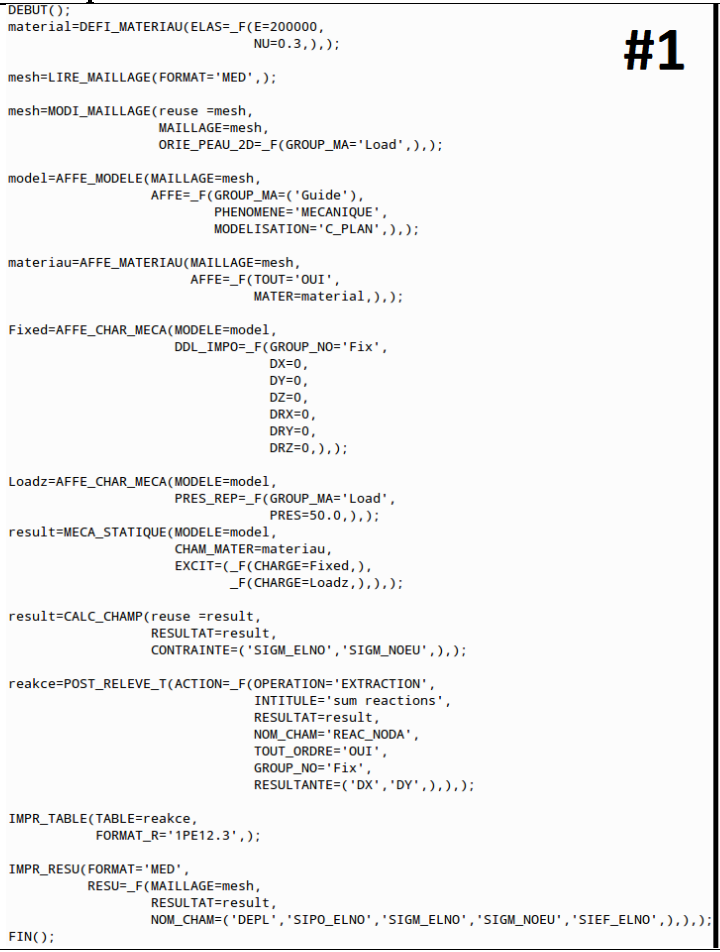
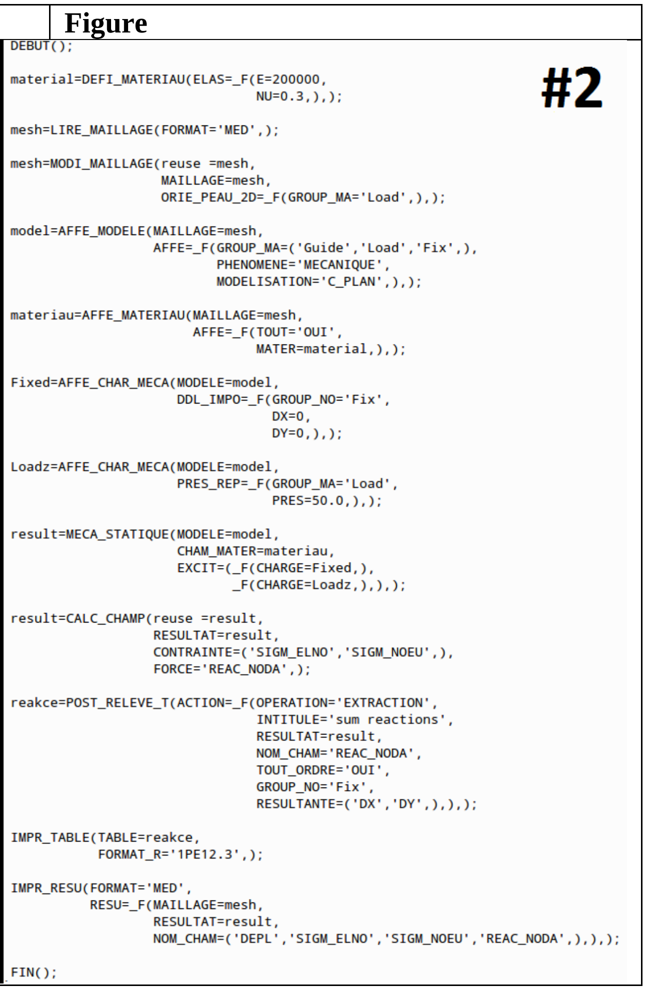
#	Description	Figure
10.	Open your command file with a text editor, and locate the part where your fixed support is. Delete manually the hightlighted orientation as we will not need them in this Analysis.	

```
Fixed=AFFE_CHAR_MECA(MODELE=model,
                     DDL_IMPO=_F(GROUP_NO='Fix',
                                 DX=0,
                                 DY=0,
                                 DZ=0,
                                 DRX=0,
                                 DRY=0,
                                 DRZ=0,),);

Loadz=AFFE_CHAR_MECA(MODELE=model,
                     PRES_REP=_F(GROUP_MA='Load',
                                 PRES=50.0,),);
```

#	Description	Figure
11.	Save the changes in the text editor and get back to Salome_Meca. Now we need to run the new Analysis again. In order to update the stream data, right-click to your study case → Edit, leave everything as they are, right-click again → Run. What we did now, is update the study case with the new modification we made.	

#	Description	Figure
12.	Apparently we got more than one code mistakes. This time, if you translate the message on the right, it says that 6 mesh points do not belong to the assigned mesh but they were assigned to the key factor PRESS_REP. To fix that we will need to get back to Eficas Application, locate AFFE_MODELE, delete GROUP_MA and assign the mesh to all(TOUT=OUI).	
13.	After running the Analysis, we will see that everything is going as they are supposed to. Post-Pro branch appears and your Object Browser should look like in the figure on your right. If you notice, there is one branch missing from the Fields group. In the IMPR_RESU command we assigned 4 different results that we want Code_Aster to present in Post-Pro module (DEPL, REAC_NODA, SIGM_NOEU and SIGM_ELNO). We are missing REAC_NODA, and that is why we did not ask CALC_CHAMP to calculate it, and IMPR_RESU to print it. Locate CALC_CHAMP and IMPR_RESU with the help of Eficas Application. Find and add FORCE from the “Mots Clefs Permis” list, and assign REAC_NODA. And do the same in RESU, IMPR_RESU concept. Update and re-run the Analysis.	
14.	The reason I am showing you all these errors is to give you a hint on how to locate your mistake, translate it, study it, find a solution, correct it, update the stream data on Salome_Meca and re-run the Analysis. Unfortunatelly you will spent most of your time finding and correcting your code mistakes. In this case we were lucky because we only made one error and it was an easy to handle, one. Below you can see how our command file was in the beginning, after we finished with Eficas Application, and how it looked after our modifications.	

```
CR d'execution de JDC en MIXTE
   !!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
   ! <S> Exception utilisateur levee mais pas interceptee.                        !
   ! Les bases sont fermees.                                                      !
   ! Type de l'exception : error                                                  !
   !                                                                              !
   !  les 6 mailles imprimées ci-dessus n'appartiennent pas au modèle et pourtant !
   ! elles ont été affectées dans le mot-clé facteur :                            !
   ! PRES_REP                                                                     !
   !!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
fin CR d'execution de JDC en MIXTE
```

#	Description	Figure
15.		

#1

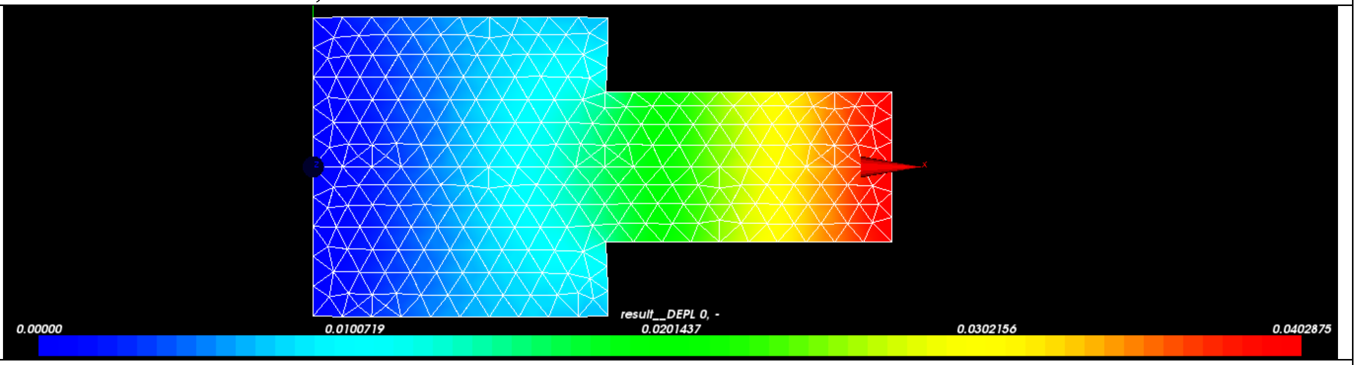
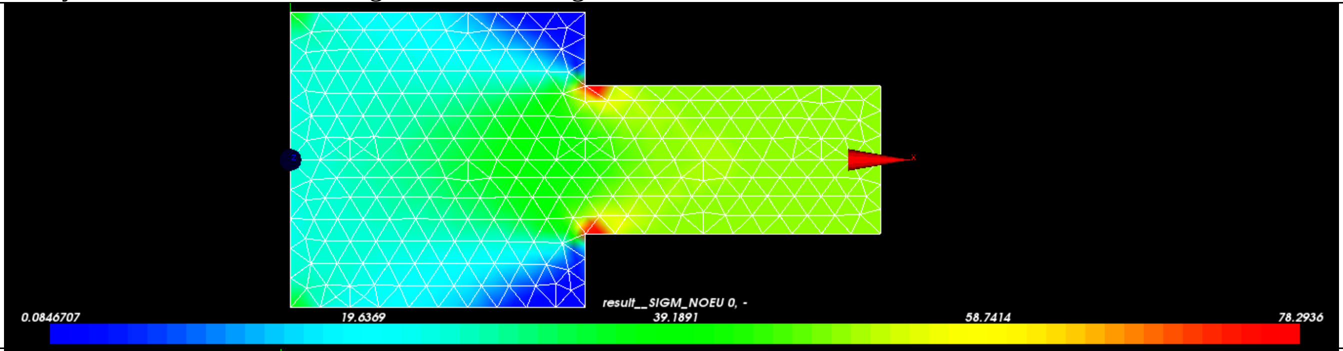
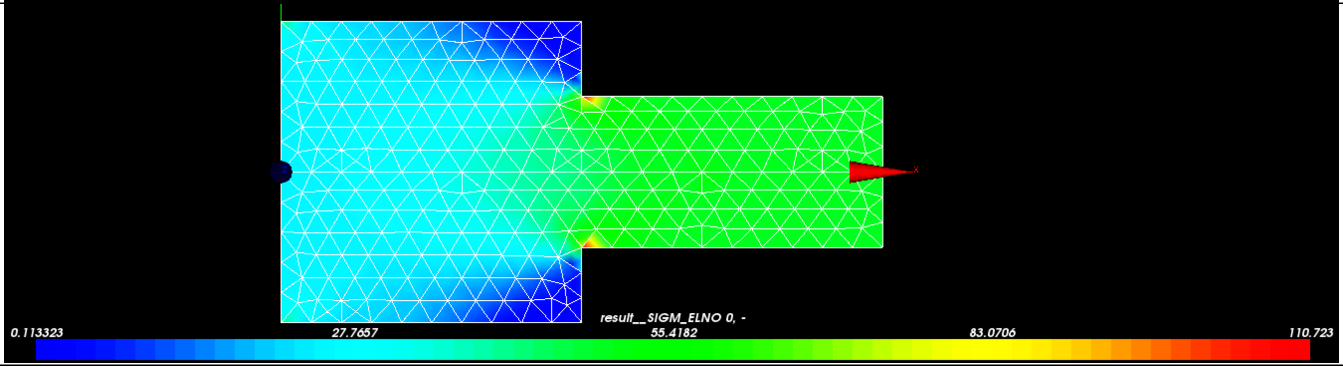
```
DEBUT();
material=DEFI_MATERIAU(ELAS=_F(E=200000,
                               NU=0.3,),);

mesh=LIRE_MAILLAGE(FORMAT='MED',);

mesh=MODI_MAILLAGE(reuse =mesh,
                   MAILLAGE=mesh,
                   ORIE_PEAU_2D=_F(GROUP_MA='Load',),);

model=AFFE_MODELE(MAILLAGE=mesh,
                  AFFE=_F(GROUP_MA=('Guide'),
                          PHENOMENE='MECANIQUE',
                          MODELISATION='C_PLAN',),);

materiau=AFFE_MATERIAU(MAILLAGE=mesh,
                       AFFE=_F(TOUT='OUI',
                               MATER=material,),);

Fixed=AFFE_CHAR_MECA(MODELE=model,
                     DDL_IMPO=_F(GROUP_NO='Fix',
                                 DX=0,
                                 DY=0,
                                 DZ=0,
                                 DRX=0,
                                 DRY=0,
                                 DRZ=0,),);

Loadz=AFFE_CHAR_MECA(MODELE=model,
                     PRES_REP=_F(GROUP_MA='Load',
                                 PRES=50.0,),);
result=MECA_STATIQUE(MODELE=model,
                     CHAM_MATER=materiau,
                     EXCIT=(_F(CHARGE=Fixed,),
                            _F(CHARGE=Loadz,),),);

result=CALC_CHAMP(reuse =result,
                  RESULTAT=result,
                  CONTRAINTE=('SIGM_ELNO','SIGM_NOEU',),);

reakce=POST_RELEVE_T(ACTION=_F(OPERATION='EXTRACTION',
                               INTITULE='sum reactions',
                               RESULTAT=result,
                               NOM_CHAM='REAC_NODA',
                               TOUT_ORDRE='OUI',
                               GROUP_NO='Fix',
                               RESULTANTE=('DX','DY',),),);

IMPR_TABLE(TABLE=reakce,
           FORMAT_R='1PE12.3',);

IMPR_RESU(FORMAT='MED',
          RESU=_F(MAILLAGE=mesh,
                  RESULTAT=result,
                  NOM_CHAM=('DEPL','SIPO_ELNO','SIGM_ELNO','SIGM_NOEU','SIEF_ELNO',),),);
FIN();
```

#2

```
DEBUT();

material=DEFI_MATERIAU(ELAS=_F(E=200000,
                               NU=0.3,),);

mesh=LIRE_MAILLAGE(FORMAT='MED',);

mesh=MODI_MAILLAGE(reuse =mesh,
                   MAILLAGE=mesh,
                   ORIE_PEAU_2D=_F(GROUP_MA='Load',),);

model=AFFE_MODELE(MAILLAGE=mesh,
                  AFFE=_F(GROUP_MA=('Guide','Load','Fix',),
                          PHENOMENE='MECANIQUE',
                          MODELISATION='C_PLAN',),);

materiau=AFFE_MATERIAU(MAILLAGE=mesh,
                       AFFE=_F(TOUT='OUI',
                               MATER=material,),);

Fixed=AFFE_CHAR_MECA(MODELE=model,
                     DDL_IMPO=_F(GROUP_NO='Fix',
                                 DX=0,
                                 DY=0,),);

Loadz=AFFE_CHAR_MECA(MODELE=model,
                     PRES_REP=_F(GROUP_MA='Load',
                                 PRES=50.0,),);

result=MECA_STATIQUE(MODELE=model,
                     CHAM_MATER=materiau,
                     EXCIT=(_F(CHARGE=Fixed,),
                            _F(CHARGE=Loadz,),),);

result=CALC_CHAMP(reuse =result,
                  RESULTAT=result,
                  CONTRAINTE=('SIGM_ELNO','SIGM_NOEU',),
                  FORCE='REAC_NODA',);

reakce=POST_RELEVE_T(ACTION=_F(OPERATION='EXTRACTION',
                               INTITULE='sum reactions',
                               RESULTAT=result,
                               NOM_CHAM='REAC_NODA',
                               TOUT_ORDRE='OUI',
                               GROUP_NO='Fix',
                               RESULTANTE=('DX','DY',),),);

IMPR_TABLE(TABLE=reakce,
           FORMAT_R='1PE12.3',);

IMPR_RESU(FORMAT='MED',
          RESU=_F(MAILLAGE=mesh,
                  RESULTAT=result,
                  NOM_CHAM=('DEPL','SIGM_ELNO','SIGM_NOEU','REAC_NODA',),),);

FIN();
```

Step 11: Post Processing the Results

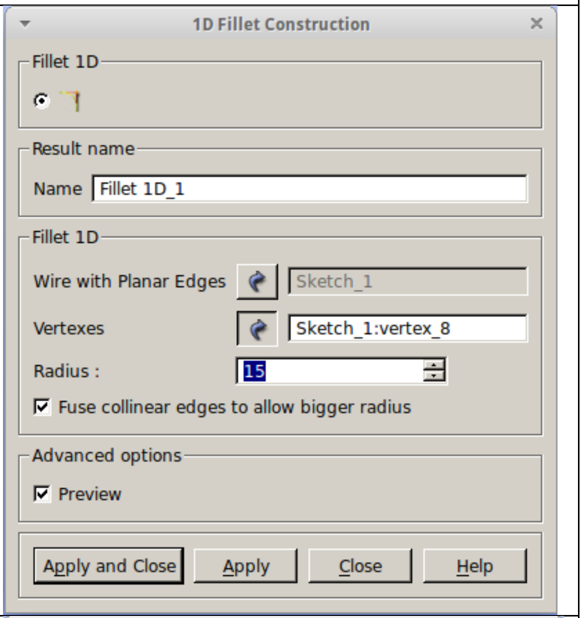
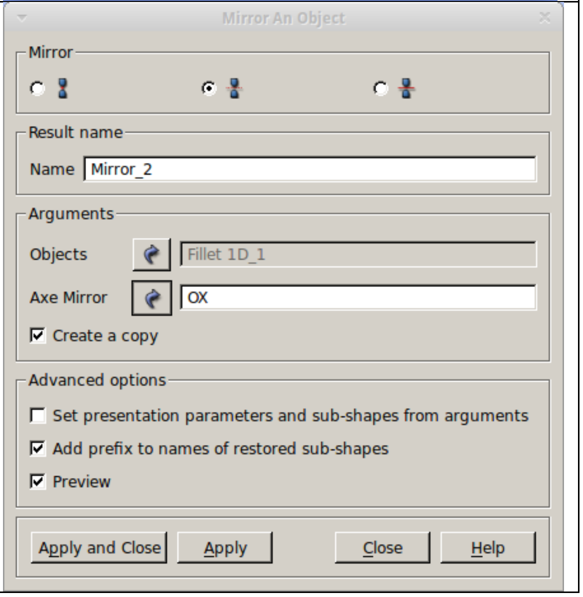
# Enable Post-Pro module and start interrogating the results of our Finite Element Analysis.

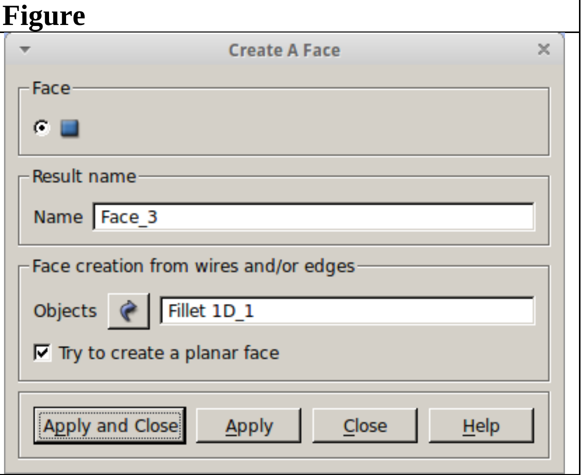
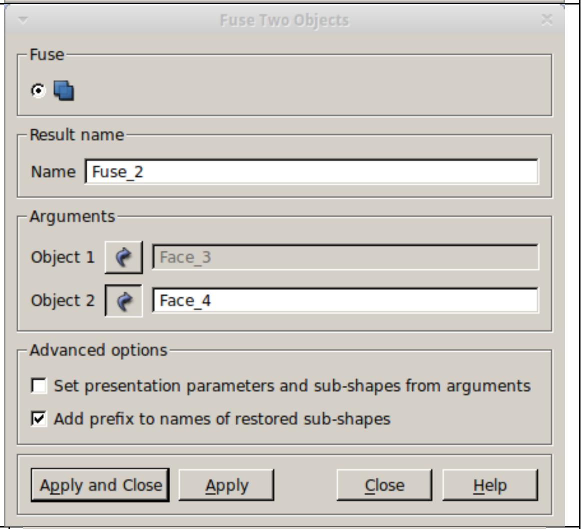
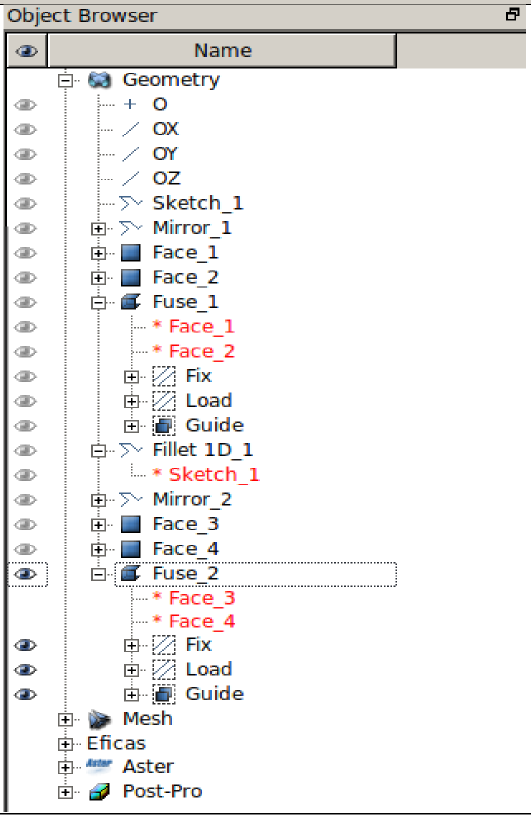
#	Description	Figure
1.	To acquire the deformation visualization, do the same steps as usual. Right-click on 0,- → Deformed Shape → Tick Magnitude Coloring and leave the rest default. Right-click on Def.Shape → Show Only → Representation → Surfaceframe. And you should have something similar to the figure below. Min. Deformation = 0 mm, Max. Deformation = 0.040 mm	
2.	 result__DEPL 0, - 0.00000 0.0100719 0.0201437 0.0302156 0.0402875	
3.	To check the Normal Averaged and Unaveraged Stress results, we have to open both result_SIGM_ELNO and result_SIGM_NOEU. Right-click on 0,- → Scalar Map → Scalar Mode = Modulus. (default) Right-click on Scalar Map → Show Only → Representation → Surfaceframe. And you should have something similar to the figures below.	
4.	 result__SIGM_NOEU 0, - 0.0846707 19.6369 39.1891 58.7414 78.2936	
5.	 result__SIGM_ELNO 0, - 0.113323 27.7657 55.4182 83.0706 110.723	
6.	SIGM_NOEU → Averaged Stress. Min. Stress = 0.08 MPa, Max. Stress = 78.3 MPa. SIGM_ELNO → Unaveraged Stress. Min. Stress = 0.11 MPa, Max. Stress = 110.7 MPa.	

Modifying the Geometry

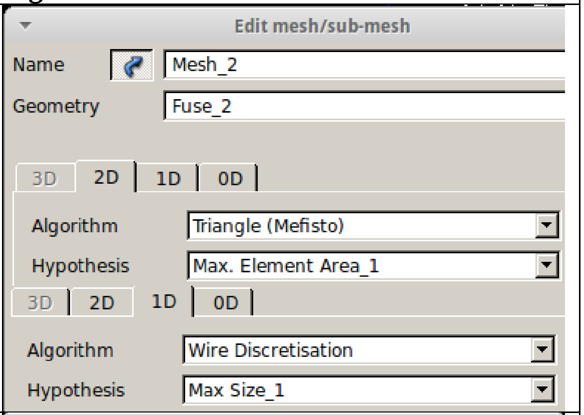
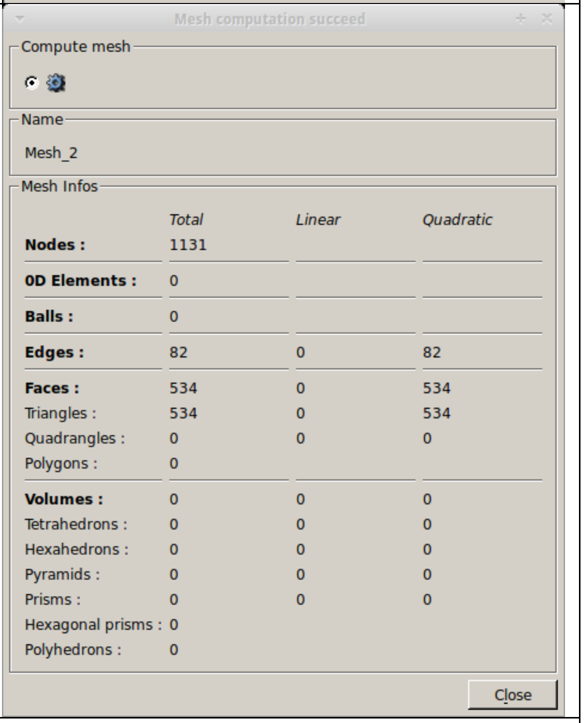
- # At this point we are going to modify our geometry. We will use fillet on the sharp edges and create a radius. In order to do that, we will need to create a whole new study. Feel free to create a new folder in whatever location you want, and name it Filleted_Bar.
- # We will use the same geometry, bar, and we are going to modify it, copy the .hdf and the .comm file and paste them into your newly created folder.

Step I: Model Geometry

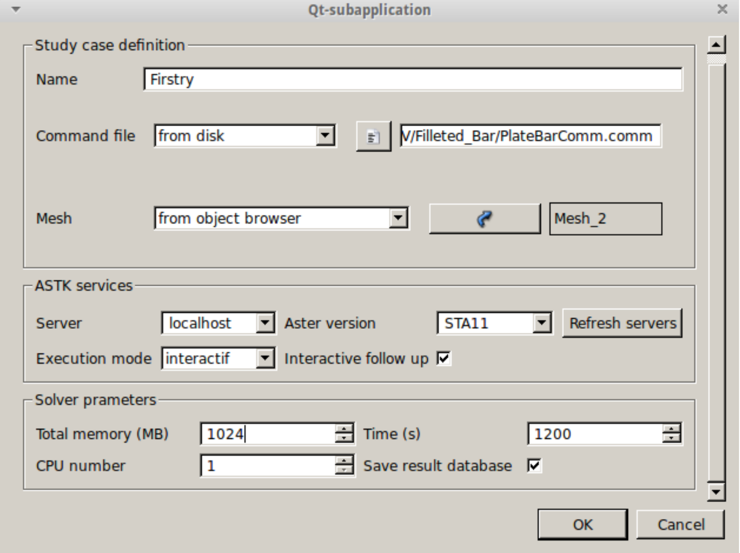
#	Description	Figure
1.	Open Salome_Meca and load your .hdf file containing the previous task (Bar). In order to modify our geometry and then follow the steps for the FE Analysis, it would be easier for you if you delete all parameters below Geometry, such us Mesh, Aster Study case.	
2.	Start-up Geometry Module. Go to Operations → Fillet_1D. In the pop-up window that appears, select Sketch_1 for “Wire with Planar Edges”. Select the vertex that connects the two lines (Sharp Corner), for “Vertexes”. Input 15mm for “Radius”. *Click “Apply and Close”*	 1D Fillet Construction Fillet 1D ● Result name Name Fillet 1D_1 Fillet 1D Wire with Planar Edges Sketch_1 Vertexes Sketch_1:vertex_8 Radius : 15 ☑ Fuse collinear edges to allow bigger radius Advanced options ☑ Preview Apply and Close Apply Close Help
3.	Now go to Operations → Transformations → Mirror Image. In the pop-up window that appears, select the 2nd Algorithm for “Mirror”. Leave default name. Choose Fillet 1D_1 for “Objects”. Choose OX Plane for “Axe Mirror”. *Finish with “Apply and Close”*	 Mirror An Object Mirror ○ ● ○ Result name Name Mirror_2 Arguments Objects Fillet 1D_1 Axe Mirror OX ☑ Create a copy Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes ☑ Preview Apply and Close Apply Close Help

#	Description	Figure
4.	Time to create the faces out of the two drawings. Go to New Entity → Build → Face. For Face_3 choose Fillet 1D_1 for "Objects" and for Face_4 select Mirror_2 for "Objects". *Finish with "Apply and Close"*	 Create A Face Face Result name Name Face_3 Face creation from wires and/or edges Objects Fillet 1D_1 ☑ Try to create a planar face Apply and Close · Apply · Close · Help
5.	Next part is to create a shape out of the two faces we assigned. Go to Operations → Boolean → Fuse as we did before. Leave default name, and assign the Arguments accordingly. Face_3 for "Object 1". Face_4 for "Object 2". *Finish with "Apply and Close"*	 Fuse Two Objects Fuse Result name Name Fuse_2 Arguments Object 1 Face_3 Object 2 Face_4 Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes Apply and Close · Apply · Close · Help
6.	Now you have created the two fillets needed for our Analysis. Save your work, be careful not to overwrite with the previous task, and let's continue to assign our groups. You don't need to delete the old groups which we assigned for Fuse_1. We will create the same groups for Fuse_2 now. Your Object Browser should look like the figure on the right.	 Object Browser Name Geometry O OX OY OZ Sketch_1 Mirror_1 Face_1 Face_2 Fuse_1 Face_1 Face_2 Fix Load Guide Fillet 1D_1 Sketch_1 Mirror_2 Face_3 Face_4 Fuse_2 Face_3 Face_4 Fix Load Guide Mesh Eficas Aster Post-Pro
7.	Save again, and open Mesh Module from the drop down list.	

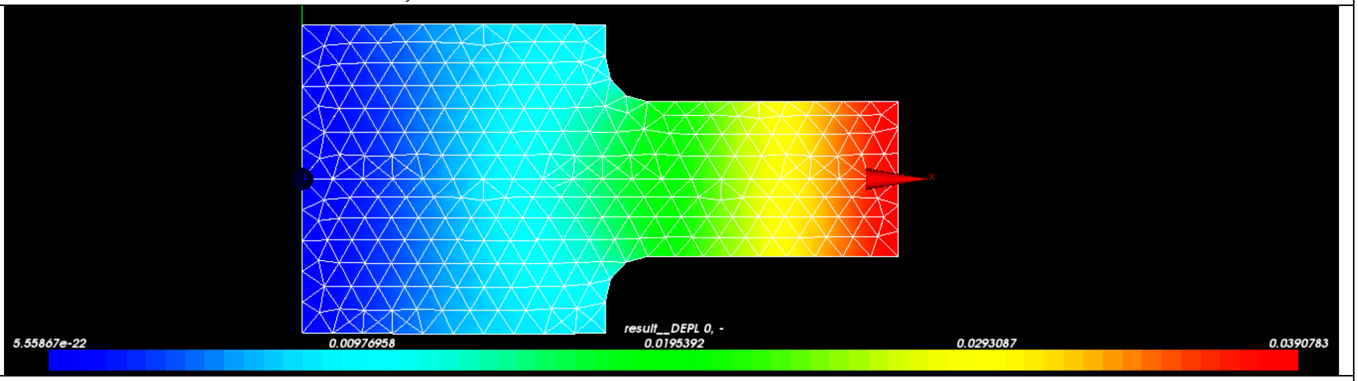
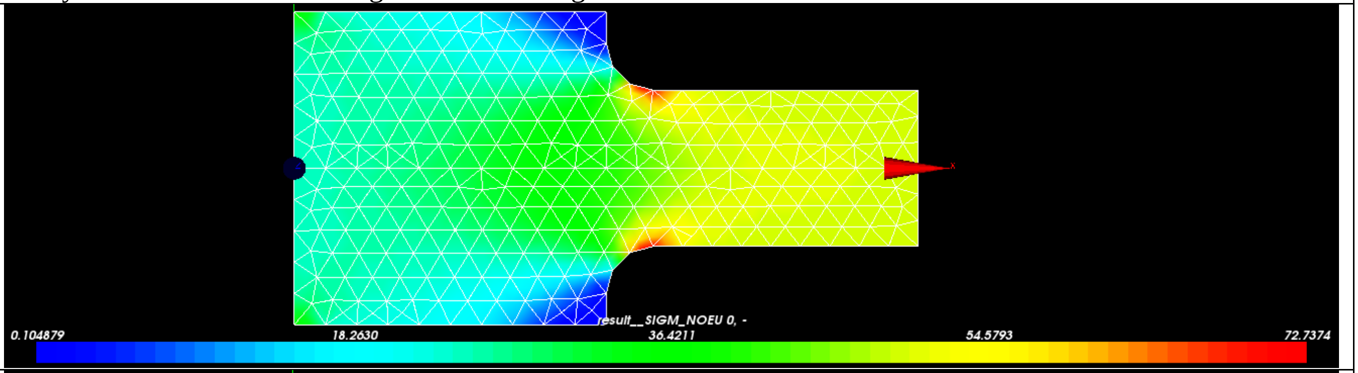
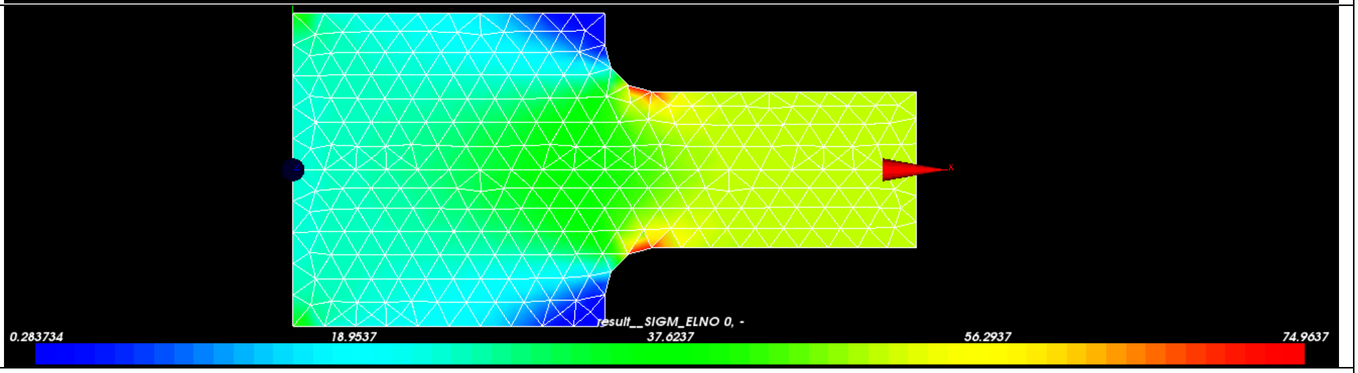
Step II: Meshing Geometry

#	Description	Figure
1.	As soon as Mesh Module loads, go to Mesh → Create Mesh, and fill in the blanks like the picture on the right. The Algorithms and the Hypothesis for both 2D and 1D tabs are the same like in the non-filleted geometry. Furthermore, Salome_Meca has stored the Hypothesis Parameters so you won't have to assign them again. *Finish with "Apply and Close"*	 Edit mesh/sub-mesh Name: Mesh_2 Geometry: Fuse_2 2D Algorithm: Triangle (Mefisto) Hypothesis: Max. Element Area_1 1D Algorithm: Wire Discretisation Hypothesis: Max Size_1
2.	Right-click on Mesh_2 → Create Groups from Geometry. Right-click on Mesh_2 → Convert to/from quadratic. Right-click on Mesh_2 → Compute. In the figure on the right, there is my mesh information. We did not do anything different than the previous Mesh_1.	 Mesh computation succeed Compute mesh Name: Mesh_2 Mesh Infos: *Total* *Linear* *Quadratic* **Nodes :** 1131 **0D Elements :** 0 **Balls :** 0 **Edges :** 82 0 82 **Faces :** 534 0 534 Triangles : 534 0 534 Quadrangles : 0 0 0 Polygons : 0 **Volumes :** 0 0 0 Tetrahedrons : 0 0 0 Hexahedrons : 0 0 0 Pyramids : 0 0 0 Prisms : 0 0 0 Hexagonal prisms : 0 Polyhedrons : 0 Close
3.	That would be all the necessary information to modify for both Geometry and Mesh Module. Let's continue now to create our command file. In this case we will not have to change anything at all to our command file, because the changes we did in the Geometry are not affecting the commands. Also we have the same groups from Geometry who also got the same names.	

Step III: Creating the command file, run the Analysis

#	Description	Figure
1.	Open up Aster Module from the drop down list. Go to Aster → Add Study Case. Fill in the gaps according to the figure on the right. Notice that from Mesh, I chose Mesh_2. Which means that even though we got the same group names in Fuse_1 and Fuse_2, the Analysis will take into consideration only the geometry which we assigned to the Mesh_2, that would be Fuse_2. *Finish with "Ok"*	 Qt-subapplication Study case definition Name: Firstry Command file: from disk, /Filleted_Bar/PlateBarComm.comm Mesh: from object browser, Mesh_2 ASTK services Server: localhost Aster version: STA11 Execution mode: interactif Interactive follow up ☑ Solver prameters Total memory (MB): 1024 Time (s): 1200 CPU number: 1 Save result database ☑
2.	Now it is time to run the Analysis. Right-click to "Firstry" (or the name you assigned for the study), select Run. Everything should work out fine, without facing any problems or errors, and Post-Pro branch should appear.	

Step IV: Post Processing the results

#	Description	Figure
1.	To acquire the deformation visualization, do the same steps as usual. Right-click on 0,- → Deformed Shape → Tick Magnitude Coloring and leave the rest default. Right-click on Def.Shape → Show Only → Representation → Surfaceframe. And you should have something similar to the figure below. Min. Deformation = 5.56e-22 mm, Max. Deformation = 0.039 mm	
2.	 result__DEPL 0, - 5.55867e-22 0.00976958 0.0195392 0.0293087 0.0390783	
3.	To check the Normal Averaged and Unaveraged Stress results, we have to open both result_SIGM_ELNO and result_SIGM_NOEU. Right-click on 0,- → Scalar Map → Scalar Mode = Modulus. (default) Right-click on Scalar Map → Show Only → Representation → Surfaceframe. And you should have something similar to the figures below.	
4.	 result__SIGM_NOEU 0, - 0.104879 18.2630 36.4211 54.5793 72.7374	
5.	 result__SIGM_ELNO 0, - 0.283734 18.9537 37.6237 56.2937 74.9637	
6.	SIGM_NOEU ➔ Averaged Stress. Min. Stress = 0.10 MPa, Max. Stress = 72.74 MPa. SIGM_ELNO ➔ Unaveraged Stress. Min. Stress = 0.28 MPa, Max. Stress = 74.96 MPa.	

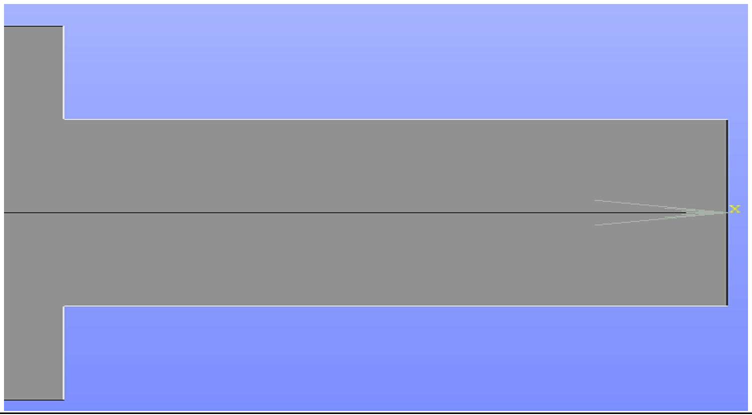
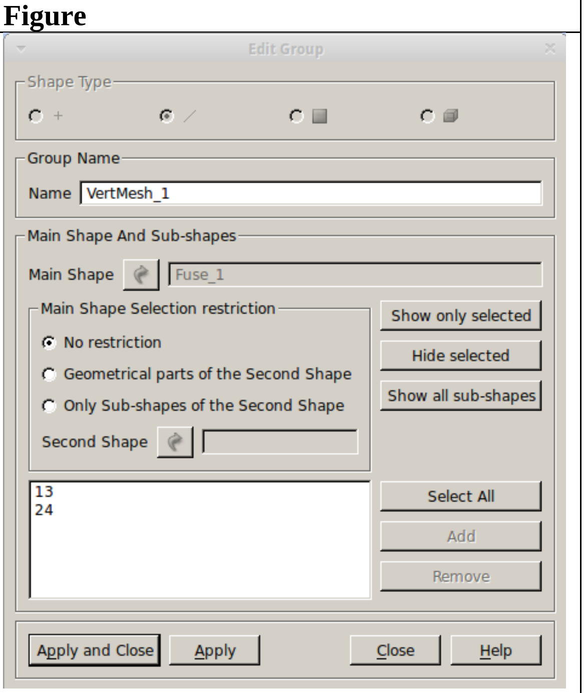
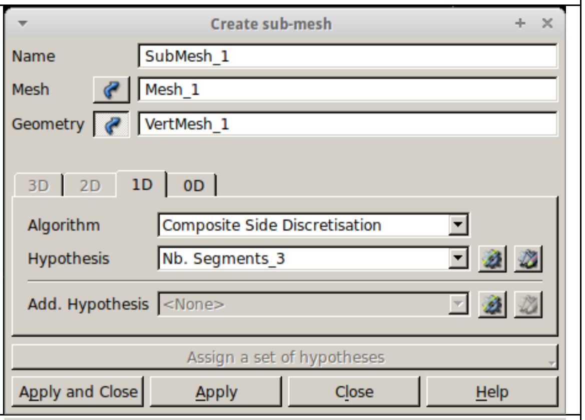
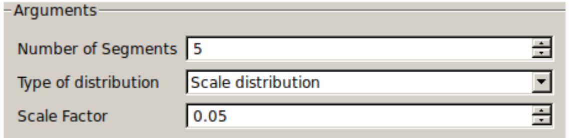
Finite Element Convergence

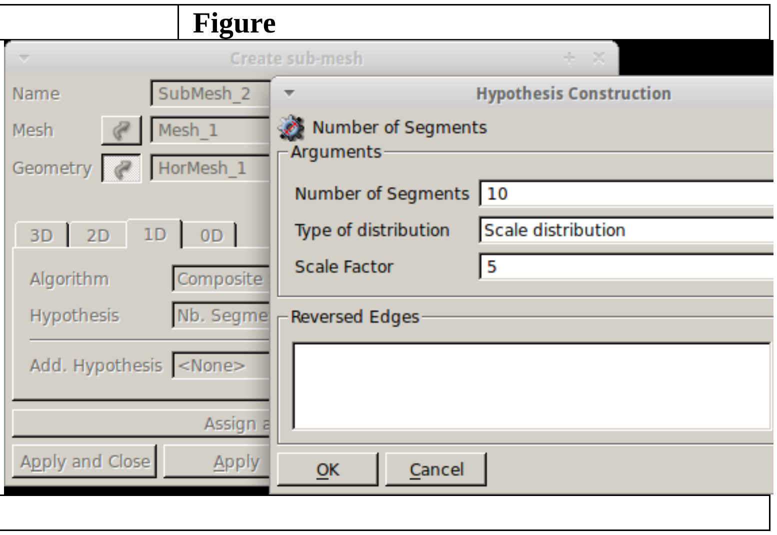
Plate Bar:

One of the core concepts of the finite element method is that, as mentioned, the finer the mesh, the more accurate the solution. Ultimately, the solution will reach the analytical solution. But how fast does it approach that solution? This is what we want to answer in this part of the chapter.

The answer depends on what kind of element we are using. In this step we will add a Sub-Mesh to our model, to make the mesh finer in a certain point of interest, and try to compare different kinds of Sub-Mesh. The comparison will of course be in the same part of the Geometry so that the results are comparable.

# To do so, we must go back to the original Plate Bar and make some modifications, after that we will modify also the Filleted Bar.

#	Description	Figure
1.	Open Salome_Meca and load the Plate Bar case study. Activate Geometry Module, in order to create some more groups to help us create the Sub-Mesh. Select Fuse_1, and go to New Entity → Group → Create. We will make two different groups, one will contain the two vertical lines, and the other one will contain the horizontal lines connected to those vertical ones. 	 Edit Group Group Name Name VertMesh_1 Main Shape Fuse_1 Main Shape Selection restriction: ● No restriction 13 24
2.	After creating the groups, activate Mesh Module from the drop down list. Go to Mesh → Create Sub-mesh, and fill the gaps according to the picture on the right. I have named VertMesh_1 and HorMesh_1 the four Vertical and Horizontal lines we are interested in making a finer mesh.	 Create sub-mesh Name SubMesh_1 Mesh Mesh_1 Geometry VertMesh_1 1D Algorithm Composite Side Discretisation Hypothesis Nb. Segments_3 Add. Hypothesis <None>
3.	This figure shows the Hypothesis Argument/settings. The aim of this section is to calculate different results by messing with the mesh. More specific, we will divide our Vertical lines to 5 (Number of Segments) elements. But we will use Scale Factor of 0.05 to distribute the elements differently between the lines length, making them finer and smaller as we reach the connection point.	 Arguments Number of Segments 5 Type of distribution Scale distribution Scale Factor 0.05

#	Description	Figure
4.	For the Horizontal lines, my Hypothesis Arguments are similar to the figure on the right. Number of Segments = 10, with Scale Factor of 5.	 Create sub-mesh Name: SubMesh_2 Mesh: Mesh_1 Geometry: HorMesh_1 Algorithm: Composite Hypothesis: Nb. Segme Add. Hypothesis: <None> Hypothesis Construction Number of Segments Arguments Number of Segments: 10 Type of distribution: Scale distribution Scale Factor: 5 Reversed Edges 3D 2D 1D 0D Assign a Apply and Close Apply OK Cancel
5.		

- # We know now how to create a Sub-mesh. I will summarize one more time, Sub-mesh gives us the ability to point out a specific area in our geometry and input different Mesh options like the ones we got in the original Mesh.
- # Finite Element Convergence is all about figuring the right Mesh and Sub-mesh, to get results as closer to reality as possible.
- # To achieve that, Mesh needs to get finer. So we will adjust the Number of Segments and the Scale Factor of the Sub-Meshes in order to analyze the results.

Results

I am using ➔ Mesh: **2D**
Algorithm → Triangle (Mefisto)
Hypothesis → Max. Element Area_1 = 20.

1D
Algorithm → Wire Discretisation
Hypothesis → Max. Size_1 = 10

[VertMesh] SubMesh_1: **1D**
Algorithm → Composite Side Discretisation
Hypothesis → Nb. Segments_1
Arguments → Number of Segments = 15
Scale Distribution
Scale Factor = 1

[HorMesh] SubMesh_1: **1D**
Algorithm → Composite Side Discretisation
Hypothesis → Nb. Segments_2
Arguments → Number of Segments = 25
Scale Distribution
Scale Factor = 20

# These are my starting configurations for the Mesh, Sub-meshes.

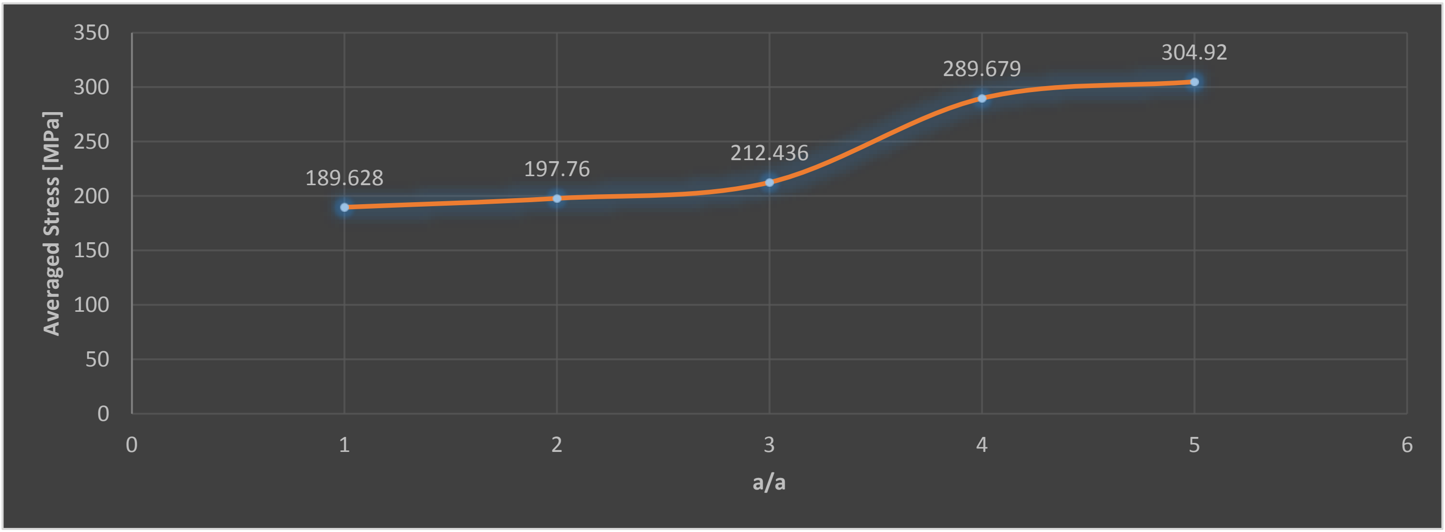
# After you have input all the Algorithms/Hypothesis/Arguments, you must now get started to calculate the given data.

# The procedure that you will follow to manually change the Arguments is actually very simple, but time consuming.

1. Activate Mesh Module.
2. Right-click one SubMesh_1 and choose Edit Mesh/Submesh.
3. Get to Hypothesis Construction, when the pop-up window appears.
4. Change the values and finish with “Apply and Close”
5. Do the same for SubMesh_2.
6. Right-click on Mesh_1 and choose Compute.
7. Right-click on Mesh_1 and choose Convert to/from quadratic.
8. Right-click on Mesh_1 and choose Compute.
9. Activate Aster Module.
10. Right-click on “assignedname” study and choose Update Mesh.
11. Right-click on “assignedname” study and choose Run.
12. Activate Post-Pro Module after the Analysis is done.
13. Get to the Averaged Stress tab, that would be result_SIGM_NOEU.
14. Assign Scalar Map with default options to 0,- .
15. Right-click on Scalar Map and choose Representation → Surfaceframe.
16. Right-click on Scalar Map and choose Show Only.

# Here are my own results.

	Nb.Seg/Scal. Fac.	*Nb.Seg/Scal. Fac.*	*Nb.Seg/Scal. Fac.*	*Nb.Seg/Scal. Fac.*	*Nb.Seg/Scal. Fac.*
a/a[1]	**1**	**2**	**3**	**4**	**5**
Sub-mesh_1	15/1	15/0.8	15/0.6	15/0.4	15/0.2
Sub-mesh_2	25/20	30/20	35/20	35/30	35/33
Averaged Stress	189.628	197.76	212.436	289.679	304.92



---

[1] 1 = Starting Mesh, 2 = Good Mesh, 3 = Very Good Mesh, 4 = Excellent Mesh, 5 = Ultimate Mesh.

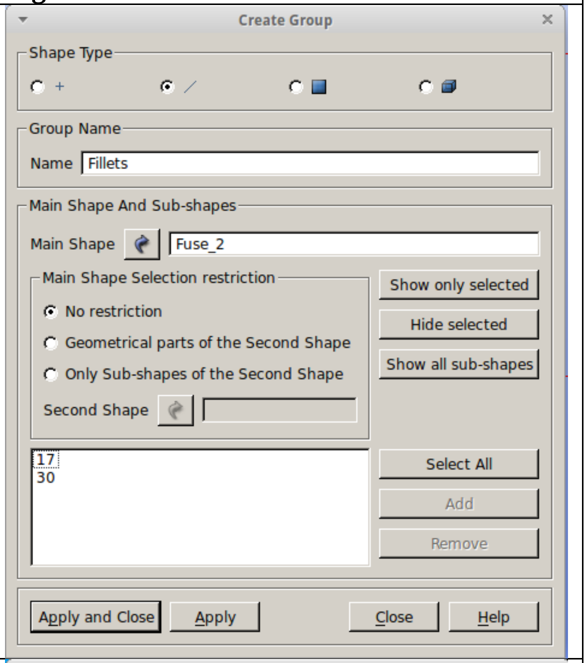
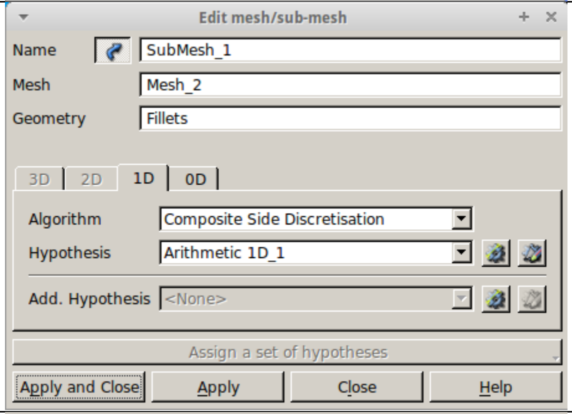
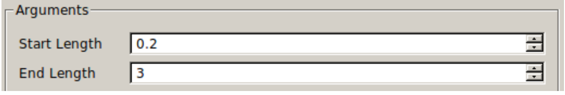
# You can see from the results of Max. Averaged Normal Stress, that from #1 to #5 (the mesh is getting finer) the difference between Stresses increases.

# For example: First Difference [197.8 - 189.6 = ~8 MPa]
Second Difference [212.4 – 197.8 = ~15 MPa]
Third Difference [289.7 – 212.4 = ~77 MPa]
Forth Difference [304.9 – 289.7 = 15 MPa]

# We can understand from those results that, the finer the mesh gets, the difference between the two results is getting bigger, theoretically touching infinite.

Filleted_Plate_Bar:

# After finishing up the simple Plate Bar, it is time to do the same to the Filleted_Plate_Bar.

#	Description	Figure
1.	Open up Salome_Meca and load the Filleted_Bar study. Activate Geometry Module. Create a group of the fillets this time, as shown in the figure on the right. *Finish with "Apply and Close"*	 Create Group Shape Type: ○ + ● ⁄ ○ ○ Group Name Name: Fillets Main Shape And Sub-shapes Main Shape: Fuse_2 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape 17 30 Show only selected Hide selected Show all sub-shapes Select All Add Remove Apply and Close Apply Close Help
2.	After creating the groups, activate Mesh Module from the drop down list. Go to Mesh → Create Sub-mesh, and fill the gaps according to the picture on the right. This time we will use Arithmetic 1D_1 for "Hypothesis". After assigning the Hypothesis don't forget to input the data for the Arguments. *After that, Finish with "Apply and Close"*	 Edit mesh/sub-mesh Name: SubMesh_1 Mesh: Mesh_2 Geometry: Fillets 3D | 2D | 1D | 0D Algorithm: Composite Side Discretisation Hypothesis: Arithmetic 1D_1 Add. Hypothesis: <None> Assign a set of hypotheses Apply and Close Apply Close Help
3.	This figure shows the Hypothesis Argument/settings. Start Length is the length of our first element. End Length is the length of the last element. The length from one element to the other is increasing automatically, starting from 0.2 up to 3.	 Arguments Start Length: 0.2 End Length: 3

# Following the same procedure like we did in the Plate_Bar, you should have your results in a couple of minutes.

Results

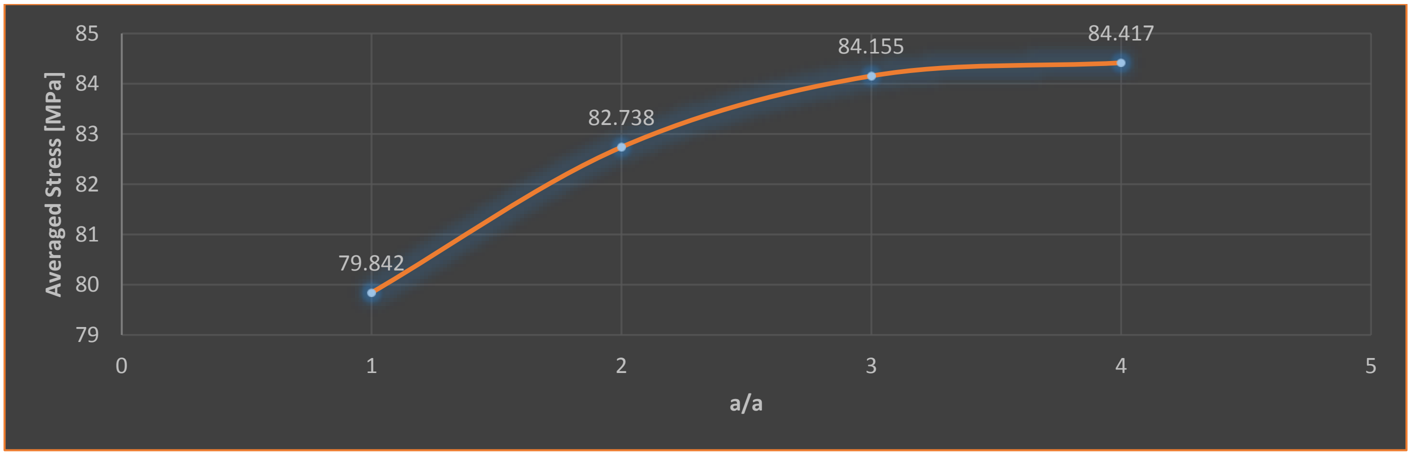
I am using ➔ Mesh: **2D**
Algorithm → Triangle (Mefisto)
Hypothesis → Max. Element Area_1 = 20.

1D
Algorithm → Wire Discretisation
Hypothesis → Max. Size_1 = 10

[Fillets] SubMesh_1: **1D**
Algorithm → Composite Side Discretisation
Hypothesis → Arithmetic 1D_1
Arguments → Start Length = 0.2
End Length = 3

# Here are my own results:

	Nb.Seg/Scal. Fac.	*Nb.Seg/Scal. Fac.*	*Nb.Seg/Scal. Fac.*	*Nb.Seg/Scal. Fac.*
a/a[2]	**1**	**2**	**3**	**4**
Sub-mesh_1	0.2 – 3	0.2 – 2	0.2 – 1	0.2 – 0.5
Averaged Stress	79.842	82.738	84.155	84.417



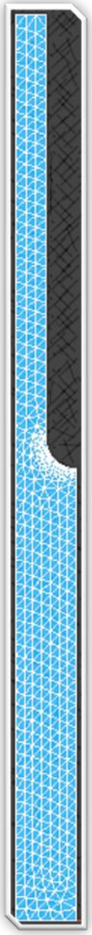
# Here, we can observe a different phenomenon than the one we saw before. In this case elements of the Sub-mesh are getting finer throughout the fillets length the Averaged Stress result difference is getting smaller, therefore convergent to some precise value.

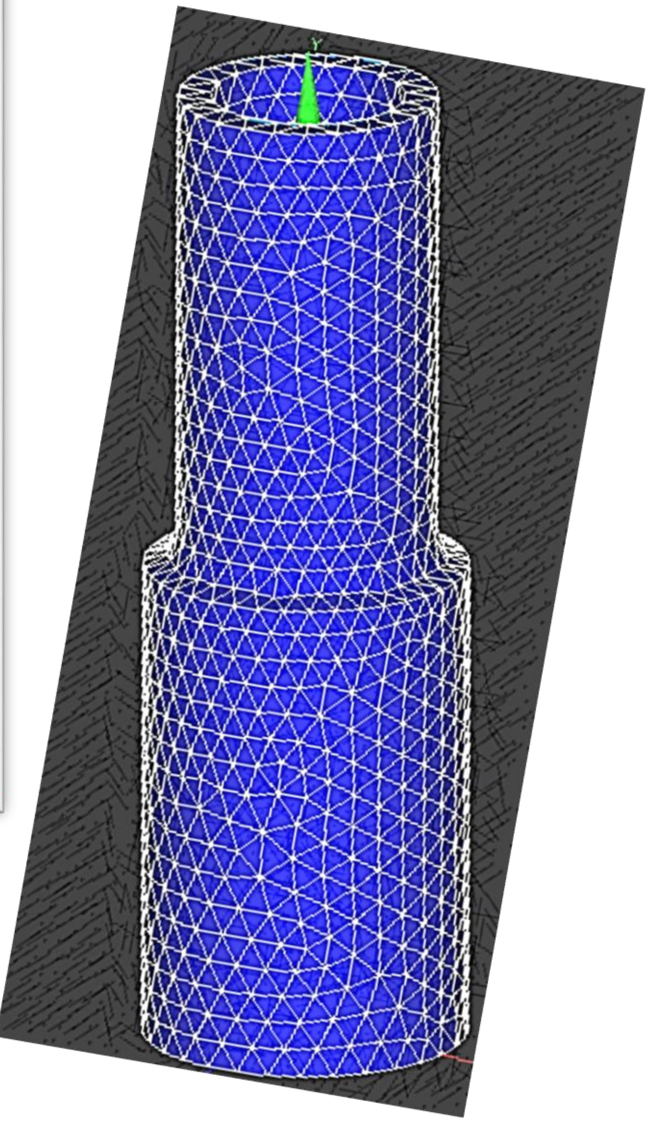
---

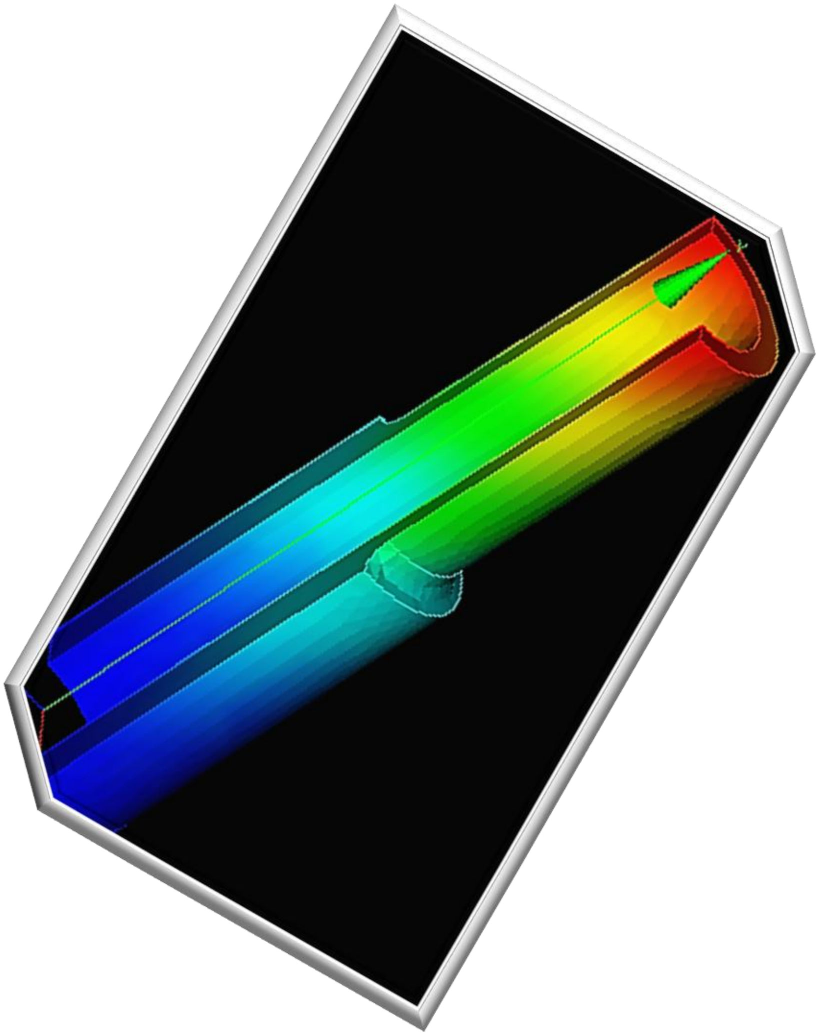
[2] 1 = Starting Mesh, 2 = Good Mesh, 3 = Very Good Mesh, 4 = Excellent Mesh, 5 = Ultimate Mesh.

Chapter_VI: Shaft

In this Chapter we are interested in a 2D Axisymmetric Analysis with a notched geometry.







Step 1: Purpose of the FE Analysis/ Description of the problem

# **Axisymmetric Analysis -** The axisymmetric problem deals with the analysis of structures of revolution under axisymmetric loading. A structure of revolution is generated by a generating cross section that rotates 360◦ about an axis of revolution. Such structures are said to be rotationally symmetric.

- # A structure of revolution by itself does not necessarily define an axisymmetric problem. It is also necessary that the loading, as well as the support boundary conditions, be rotationally symmetric.
- # Axisymmetric elements are 2D elements that can be used to model axisymmetric geometries with axisymmetric load. In simpler words, we are converting a 3D Geometry to a 2D Geometry making the model smaller, therefore faster execution and faster post processing.
- # We only model the cross section, and Salome_Meca accounts for the fact that it is really a 3D, axisymmetric structure.
- # The axisymmetric problem deals with the analysis of structures of revolution under axisymmetric loading. A structure of revolution is generated by a generating cross section that rotates 360◦ about an axis of revolution.
- # In this Chapter we consider the finite element discretization of axisymmetric solids. We were given a Steel Shaft with dimensions which are given below. Purpose of this chapter is to fully understand the concept of Axi-symmetry and its possibilities.

Reference --> *University of Wisconsin-Madison, Department of Engineering Physics.*

Examples before starting our model

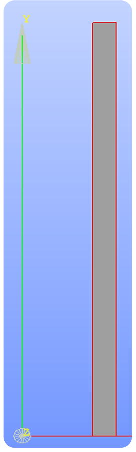
# In Salome_Meca, axisymmetric models must be drawn in OY Plane.

# The OX direction is the radial direction.

# The 2D model will be rotated about the Y axis (always about x=0).

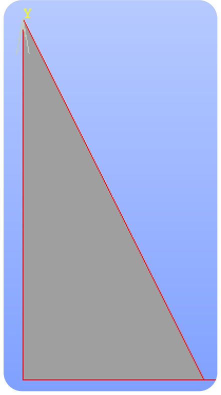
# Nothing in your model should be in the region X<0.

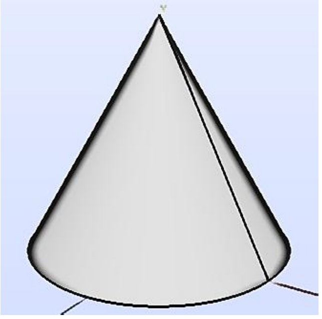
# In post processing σ_x will be the radial stress, σ_y will be the axial stress, and σ_z will be the “hoop” stress.

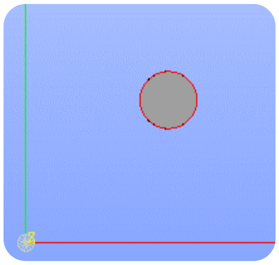


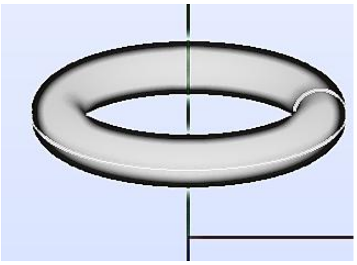
← Surfaces Needed.

Models →





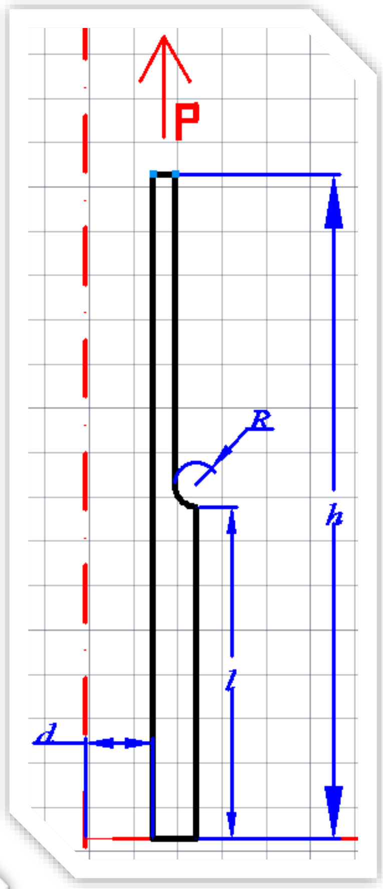


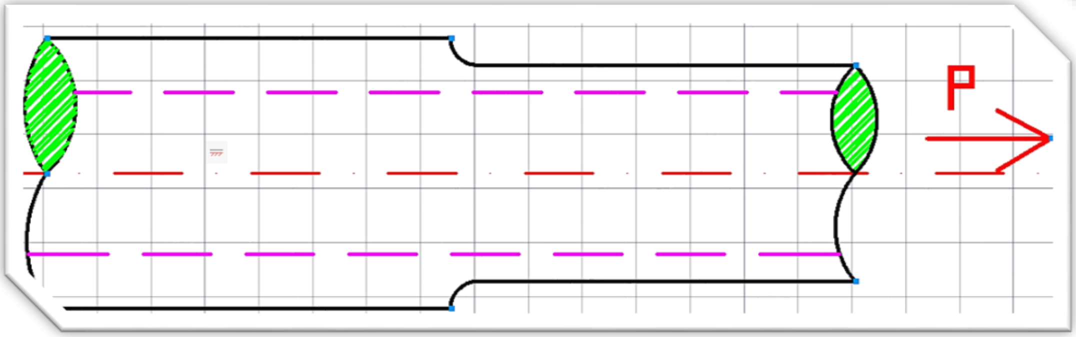


Step 2: Input Values for the FE Analysis

Shaft Dimensions: Length = 150 mm [h]; Length = 15 mm [d];
Length = 75 mm [l];
Radius = 5 mm [R];

Young's Modulus: 200 GPa;
Poisson's Ratio: 0.3;
Pressure Applied: 50 MPa [P];

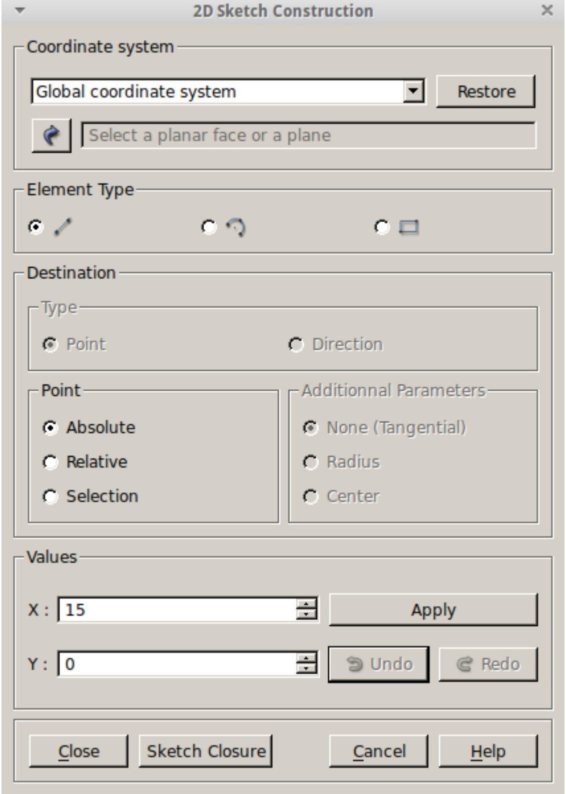
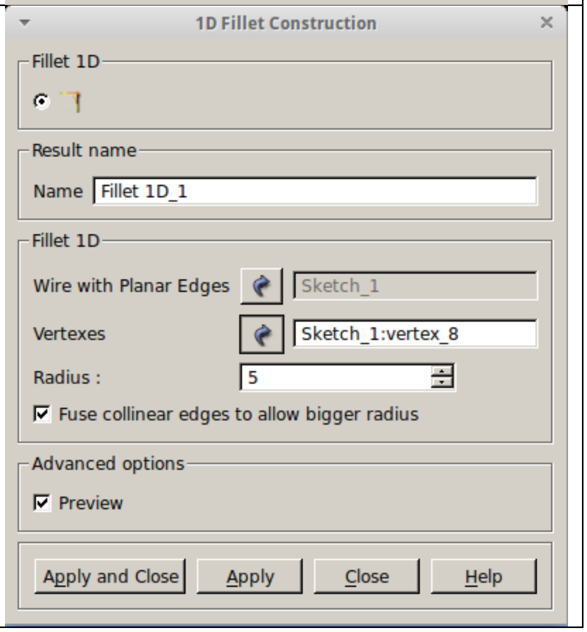


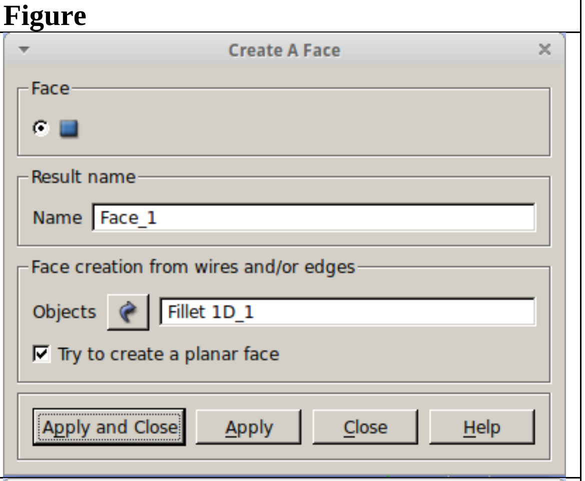
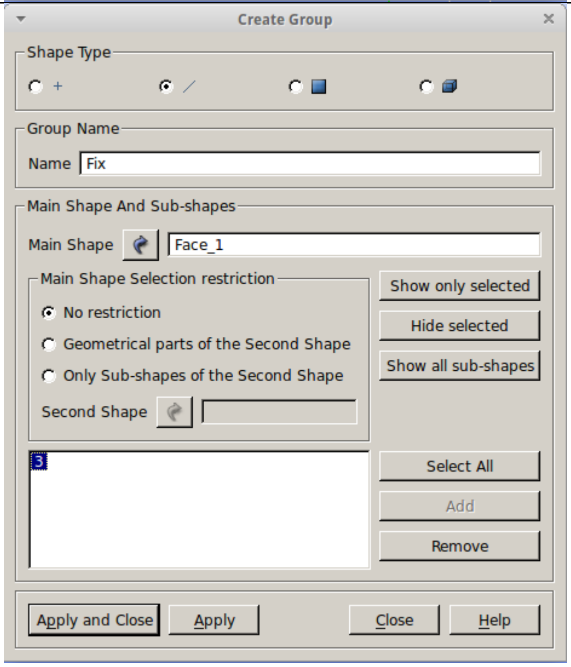
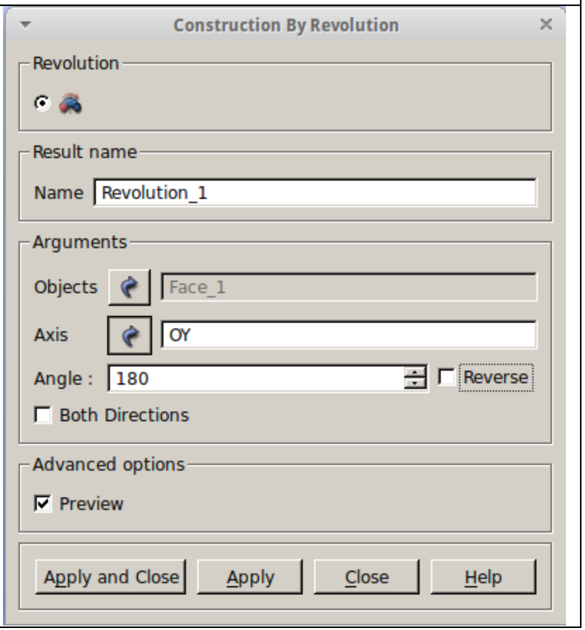


Step 3: Model Geometry

# Start Salome_Meca, create a new document, save as “Shaft”, activate Geometry Module from the drop down list and start sketching.

# For generating an Axi-symmetric model, we just need to create a planar face of the meridian cross section of the shaft.

#	Description	Figure
1.	In the Geometry Module, go to New Entity → Basics → 2D Sketch. These are the points we are going to use to create our 2D geometry. Point_1: (15,0) Point_2: (15,150) Point_3: (20,150) Point_4: (20,75) Point_5: (25,75) Point_6: (25,0) *Finish with “Sketch Closure”*	 2D Sketch Construction Coordinate system Restore Select a planar face or a plane Element Type Destination Type Point Direction Additionnal Parameters None (Tangential) Radius Center Relative Selection Values Apply Undo Redo Close Sketch Closure Cancel Help Coordinate system: Global coordinate system Point: Absolute X : 15 Y : 0
2.	After creating our 2D sketch, it is time to create the radius. Go to Operations → Fillet 1D. Select with the mouse the correct Vertex and input 5mm for Radius, according to the figure on the right. *Finish with “Apply and Close”*	 1D Fillet Construction Fillet 1D Result name Fillet 1D Advanced options Apply and Close Apply Close Help Name: Fillet 1D_1 Wire with Planar Edges: Sketch_1 Vertexes: Sketch_1:vertex_8 Radius : 5 ☑ Fuse collinear edges to allow bigger radius ☑ Preview

#	Description	Figure
3.	Next step is creating a face out of the sketch, in which we will assign the groups later on. Go to New Entity → Build → Face. Select Fillet 1D_1 for Object. *Finish with "Apply and Close"*	 Create A Face Face: ● Result name — Name: Face_1 Face creation from wires and/or edges — Objects: Fillet 1D_1 ☑ Try to create a planar face
4.	After creating a face, we are going to assign the groups that we are going to need later on. Go to New Entity → Group → Create. Select 2nd Algorithm for Lines, name the first group "Fix", and choose for Main Shape "Face_1". Select the Line where we want our Fixed Support to be, and press "Add". Do the same for the load. Create a group named "Load" with the same Line Algorithm and "Add" the Line where our loads is. Also you can create one more group for the fillet, if you want to make the mesh in that part finer. Now for the "Guide" group, select 3rd Algorithm for faces. "Add" the only face available to you. *Finish with "Apply and Close"*	 Create Group Shape Type: ○ + ● / ○ ○ Group Name — Name: Fix Main Shape And Sub-shapes — Main Shape: Face_1 Main Shape Selection restriction: ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape: 3 Show only selected Hide selected Show all sub-shapes Select All Add Remove Apply and Close Apply Close Help
5.	***Off-Topic.*** Another cool ability of Salome_Meca is the Revolution option. In this task we were asked to create a face which if you revolve it 360° it would turn into a shaft. That is what the Revolution command does. Go to New Entity → Generation → Revolution. Input the data like the figure on the right. *Finish with "Apply and Close".* And you can see now in the Viewer tab, your half-Shaft.	 Construction By Revolution Revolution: ● Result name — Name: Revolution_1 Arguments — Objects: Face_1 Axis: OY Angle : 180 ☐ Reverse ☐ Both Directions Advanced options — ☑ Preview

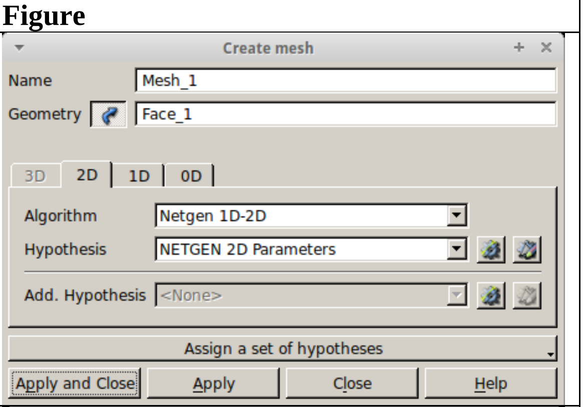
Step 4: Meshing of Geometry

# After saving your geometry study, activate Mesh Module from the drop down list and start meshing your model.

<table>
<tr><th>#</th><th>Description</th><th>Figure</th></tr>
<tr><td><i>1.</i></td><td>Highlight Face_1, and go to Mesh → Create Mesh.

Leave default name “Mesh_1” and fill in the blanks according to the figure on the right.

For Hypothesis parameters I applies for Max. Element Size= 3, and Min. Element Size= 2.

<i>Finish with “Apply and Close”</i></td><td>
Create mesh
Name Mesh_1
Geometry Face_1
2D
Algorithm Netgen 1D-2D
Hypothesis NETGEN 2D Parameters
Add. Hypothesis <None>

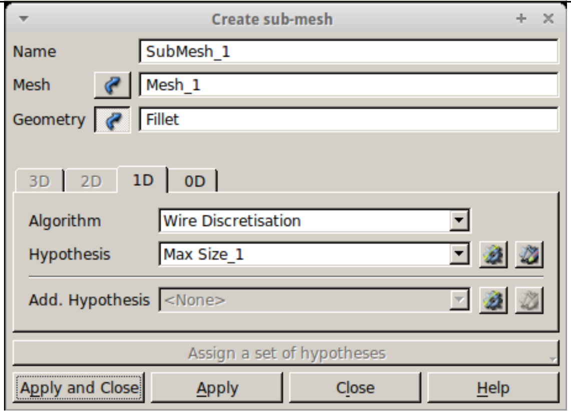
3D 2D 1D 0D
Assign a set of hypotheses
Apply and Close
Apply
Close
Help

</td></tr>
<tr><td><i>2.</i></td><td>Next part is creating a Sub-Mesh in the filleted area.

Go to Mesh → Create Sub-Mesh.

Fill in the gaps according to the figure on the right. BUT be careful for Geometry input you have to choose the group we made before that includes the fillet edge.

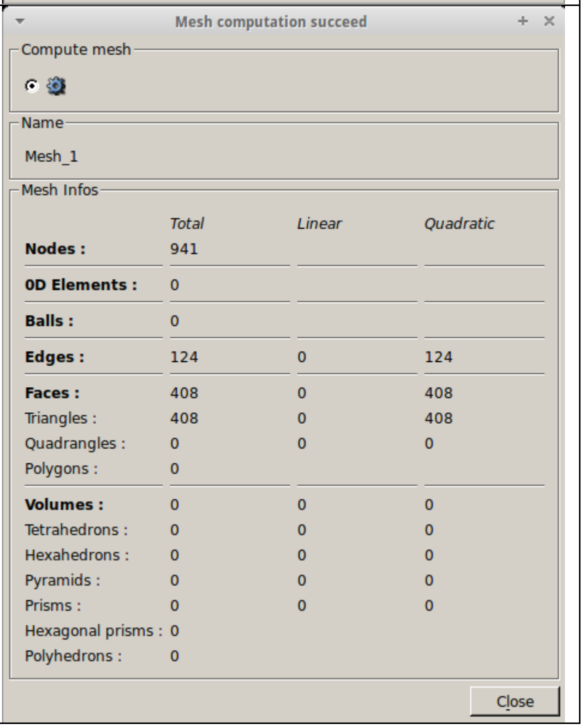
Max. Element Size = 0.5

<i>Finish with “Apply and Close”</i></td><td>
Create sub-mesh
Name SubMesh_1
Mesh Mesh_1
Geometry Fillet
1D
Algorithm Wire Discretisation
Hypothesis Max Size_1
Add. Hypothesis <None>

3D 2D 1D 0D
Assign a set of hypotheses
Apply and Close
Apply
Close
Help

</td></tr>
<tr><td><i>3.</i></td><td>After computing the Mesh_1 which includes SubMesh_1, right click on Mesh_1 and choose “Convert to/from Quadratic”.

Right-click on Mesh_1 again and choose compute.

The figure on the right shows my Mesh Information.</td><td>
Mesh computation succeed
Compute mesh
Name
Mesh_1
Mesh Infos
<table>
<tr><th></th><th><i>Total</i></th><th><i>Linear</i></th><th><i>Quadratic</i></th></tr>
<tr><td>Nodes :</td><td>941</td><td></td><td></td></tr>
<tr><td>0D Elements :</td><td>0</td><td></td><td></td></tr>
<tr><td>Balls :</td><td>0</td><td></td><td></td></tr>
<tr><td>Edges :</td><td>124</td><td>0</td><td>124</td></tr>
<tr><td>Faces :</td><td>408</td><td>0</td><td>408</td></tr>
<tr><td>Triangles :</td><td>408</td><td>0</td><td>408</td></tr>
<tr><td>Quadrangles :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Polygons :</td><td>0</td><td></td><td></td></tr>
<tr><td>Volumes :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Tetrahedrons :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Hexahedrons :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Pyramids :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Prisms :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Hexagonal prisms :</td><td>0</td><td></td><td></td></tr>
<tr><td>Polyhedrons :</td><td>0</td><td></td><td></td></tr>
</table>
</td></tr>
</table>

Step 5, 6, 7, 8, 9, 10: Salome_Meca Linear Static Analysis Wizard, Running the Analysis

# After Meshing your Geometry, it’s time to create the command file. Activate Aster Module from the drop down list and click on Aster → Wizards → Linear Elastic.

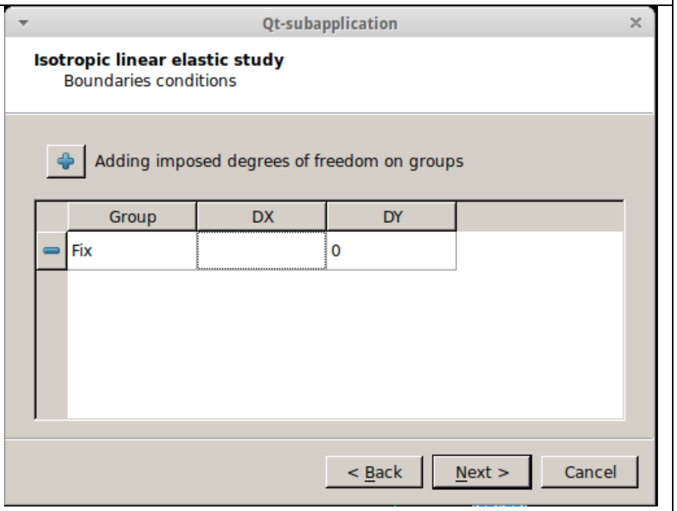
<table>
<tr><th>#</th><th>Description</th><th>Figure</th></tr>
<tr><td><i>1.</i></td><td colspan="2">In the model definition window that pops up select “Axis symmetric” and click “Next”.

Select “Mesh_1” in the next step if not already selected and click “Next”.

Enter 2e5 for Young’s Modulus and 0.3 for Poisson’s Ratio and click “Next”.</td></tr>
<tr><td><i>2.</i></td><td>For the Geometric Boundary Conditions, select “Fix” for the Group tab.

Delete “0” from “DX” as we do not want to constrain the Shaft in X direction.

For “DY” leave it default as we want it to not move in Y direction.

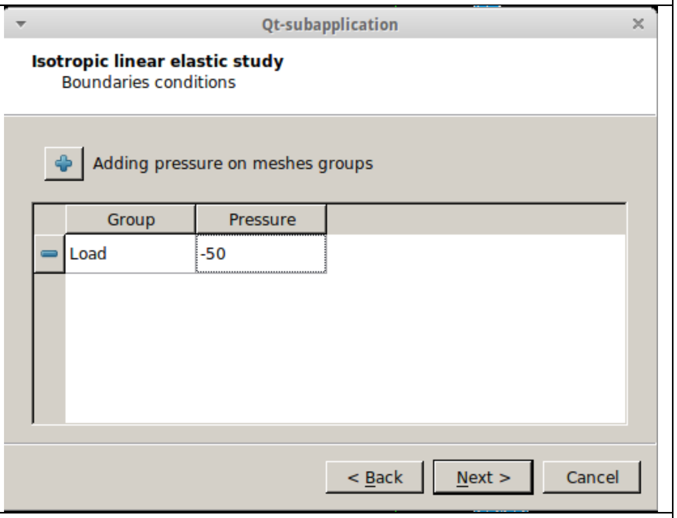
<i>Proceed with “Next”</i></td><td>

Isotropic linear elastic study
Boundaries conditions
Adding imposed degrees of freedom on groups
<table><tr><th>Group</th><th>DX</th><th>DY</th></tr><tr><td>Fix</td><td></td><td>0</td></tr></table> </td></tr>
<tr><td><i>3.</i></td><td>For the Load Boundary Conditions, select “Load” for the Group tab.

Enter “-50” as “Pressure”.

<i>Proceed with “Next”</i>

Enter a file name, and save the command file to the same folder you saved your geometry study.

<i>Finish with “Finish” option in the wizard window.</i></td><td>

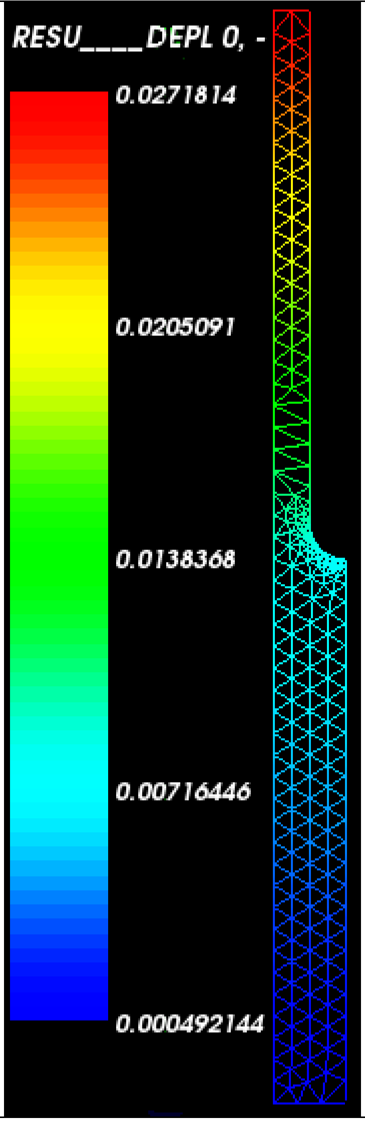
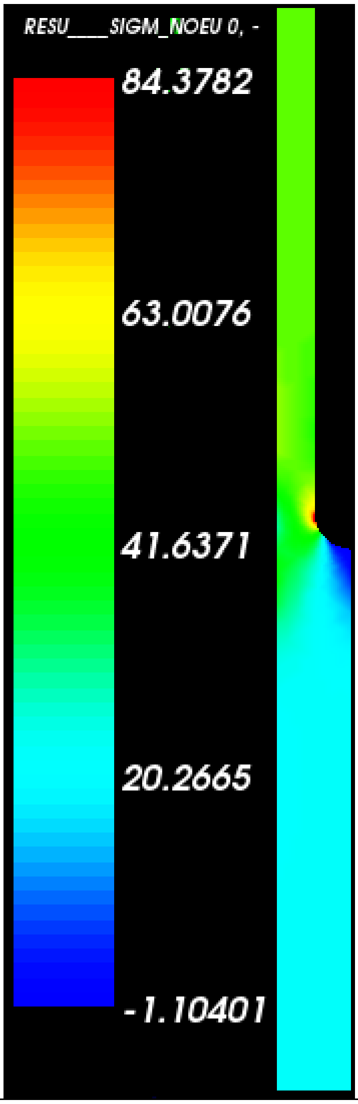
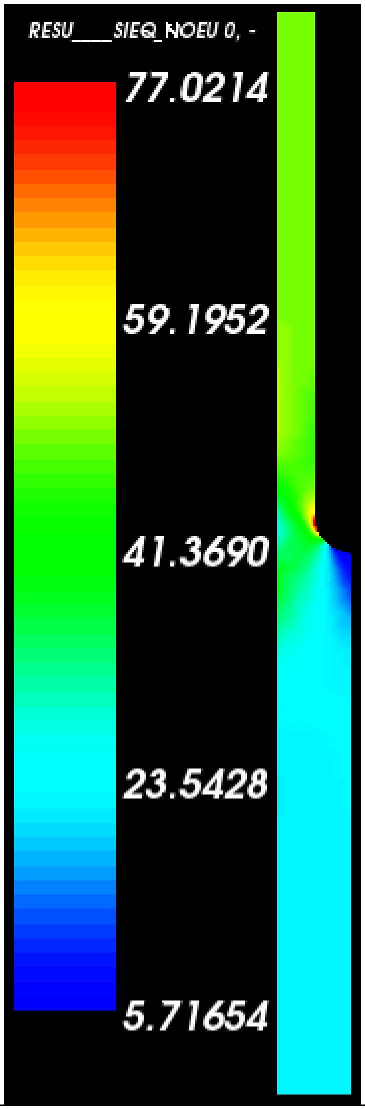
Isotropic linear elastic study
Boundaries conditions
Adding pressure on meshes groups
<table><tr><th>Group</th><th>Pressure</th></tr><tr><td>Load</td><td>-50</td></tr></table> </td></tr>
<tr><td><i>4.</i></td><td colspan="2">Check that a new branch of “Aster” has been added in the Object Browser. This concludes setting up the study for Finite Element Analysis with Salome_Meca Wizard.

Right-click on “linear-static” branch in Aster and select “Run”.

If everything goes well a new branch of “Post-Pro” will be added to the Object Browser.</td></tr>
</table>

Step 11: Post Processing of the Results

# After running the analysis and Post-Pro branch appears, it's time to analyze the results. Activate Post-Pro Module from the drop down list and

#	Description	Figure
1.	Open branch "RESU___DEPL" and right click on "0,-". From the pop up window select "Deformed Shape" and visualize it. To view how much it has deformed, right-click on "Mesh_1" in the Mesh Module without activating it and select "Show". Do the same for branch "RESU___SIGM_NOEU" and "RESU___SIEQ_NOEU". For the Normal stress (SIGM) in the pop up window change Scalar Mode to "SIYY". (Y axis direction for axi-symmetry) For the Equivalent stress (SIEQ) in the pop up window change Scalar Mode to "VMIS" I have placed my own results in the figures below.	
2.	 RESU____DEPL 0, - 0.0271814 0.0205091 0.0138368 0.00716446 0.000492144  RESU___SIGM_NOEU 0, - 84.3782 63.0076 41.6371 20.2665 -1.10401	 RESU___SIEQ_NOEU 0, - 77.0214 59.1952 41.3690 23.5428 5.71654

Stress Concentration Factor

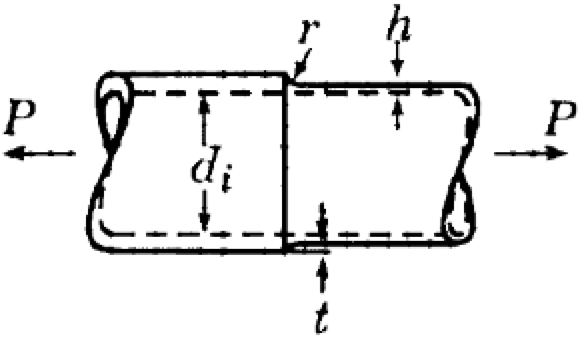
- # A stress concentration is a location in an object where stress is concentrated. An object is strongest when force is evenly distributed over its area, so a reduction in area, e.g. caused by a crack, results in a localized increase in stress. A material fail, via a propagating crack, when a concentrated stress exceeds the material's theoretical cohesive strength. The real fracture strength of a material is always lower than the theoretical value because most materials contain small cracks or contaminants that concentrate stress.
- # The stress concentrators are geometrical irregularities that cause an increase in the average effort that should be present in regions near these discontinuities, the relationship between the maximum stress that occurs and the average effort that should occur is defined as stress concentration factor; which is determined by experimental or analytical methods and presented in graphical form for ease interpretation.
- # The stress concentration factor for a tube in tension with fillet, our case, can be determined as the relation of the maximum normal stress in the discontinuity and the nominal stress, and is obtained through the equation:

$$K_t = \frac{\sigma_{max}}{\sigma_{nom}} \quad ①$$

$$(d_i/h + d_i/t) > 28 \quad ③$$

Reference --> *Pickey W.: Peterson's Stress Concentration Factors.*

$$\sigma_{nom} = \frac{P}{\pi h(d_i + h)} \quad ②$$



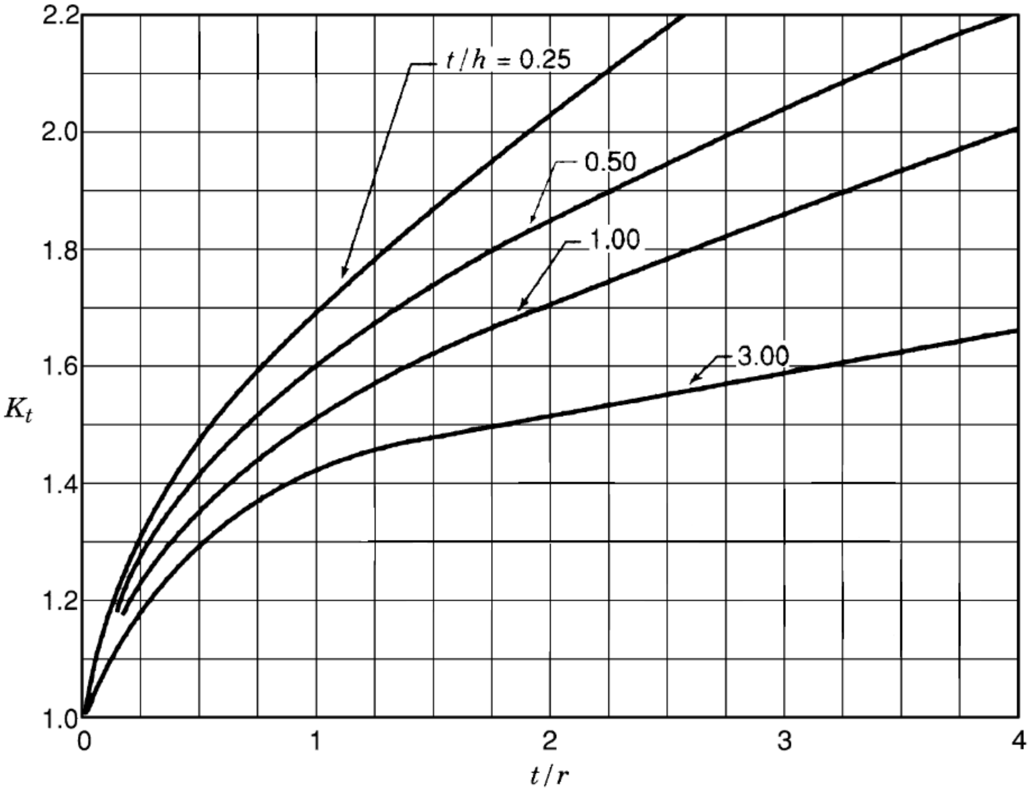


Chart Stress concentration factors K_t for a tube in tension with fillet (Lee and Ades 1956; ESDU 1981).

Hand Calculations VS Computational Calculations of Stress Concentration

Hand Calculations

As we can see in the graph, there are 2 parameters that are taken into consideration to acquire stress concentration factor. First parameter would be to determine which line do we need to choose, to accomplish that we have to solve the equation t/h with our given values. Second parameter would be to acquire the correct output for the X axis, solving the equation t/r .

According to the graph, my results are:

- **$t / r = 5/5 = 1$**
- **$t / h = 5/5 = 1$**

- **$K_t \approx 1.52$**

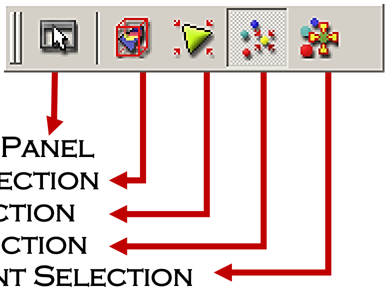
Computational Calculations

To acquire the Stress Concentration Factor for the computational calculations we are going to need Max. Normal Stress (SIGM_NOEU → SIYY) and Nominal Stress, which in our case equals to Pressure = 50MPa.

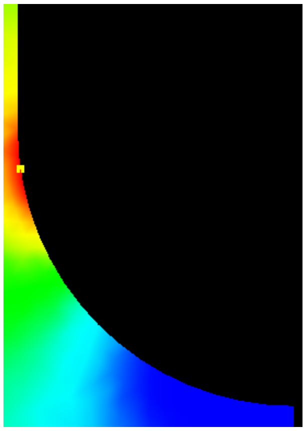
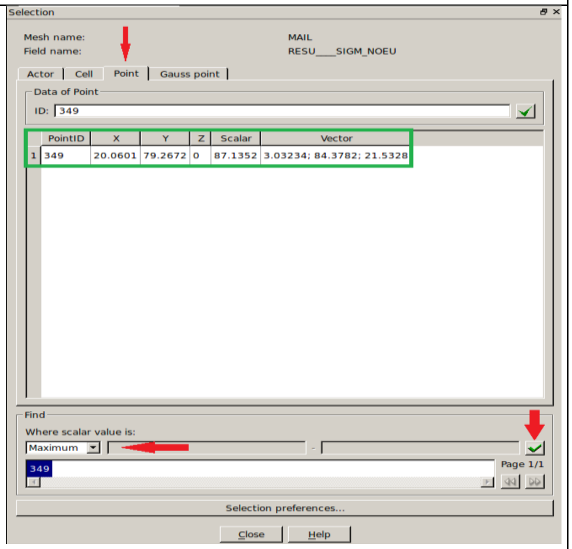
We don't need to calculate the Max. Normal Stress since Salome_Meca has an option of showing it to us.

<table>
<tr><th>#</th><th>Description</th><th>Figure</th></tr>
<tr><td><i>1.</i></td><td>This is a good opportunity to check another aspect of Salome_Meca's Post-Pro Module, an easy way to pick points throughout a geometry.

Activate Post-Pro Module from the drop down list, open RESU___SIGM_NOEU branch and highlight "Scalar Map:1".

In the top right corner, there is a group of 5 icons, identical with the ones shown in the figure on the right.
<i>Activate "Selection Panel".</i></td><td> </td></tr>
<tr><td><i>2.</i></td><td>When you activate "Selection Panel", it will open up a new window on the right side of Salome Platform.

That window has 4 main tabs. [Actor, Cell, Point, Gauss Point].

The tab that we are interested in this case is the "Point" tab.
By choosing "Point" tab and by pointing out that we are interested for the point where the Scalar Value is "Maximum", the software will give you all the information regarding the Maximum Scalar Value Point which in our case is also the Maximum Normal Stress Point.
</td><td>Selection
Mesh name: MAIL
Field name: RESU___SIGM_NOEU
Data of Point
ID: 349
<table><tr><th></th><th>PointID</th><th>X</th><th>Y</th><th>Z</th><th>Scalar</th><th>Vector</th></tr><tr><td>1</td><td>349</td><td>20.0601</td><td>79.2672</td><td>0</td><td>87.1352</td><td>3.03234; 84.3782; 21.5328</td></tr></table>Find
Where scalar value is: Maximum
349
Page 1/1 </td></tr>
<tr><td><i>3.</i></td><td colspan="2">According to the computational results: $K_t = {}^{84.3782}/_{50}$ => $K_t \approx 1.69$</td></tr>
</table>

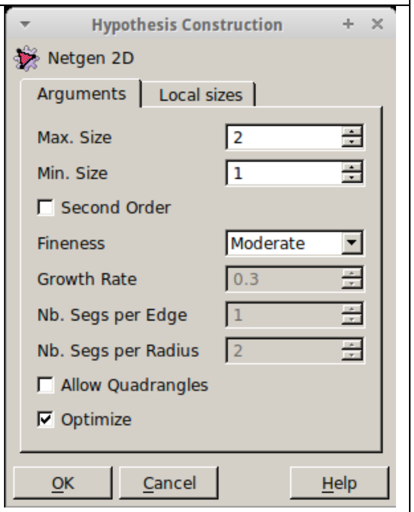
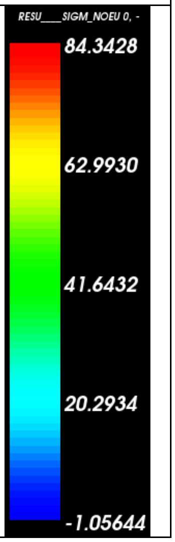
Solving the Equation

- $(d_i/h + d_i/t) > 28 \;=>\; (30/5 + 30/5) > 28 \;=>\; 12 > 28$ ③

Conclusion: As we can see the results between Hand and Computational Calculations are almost the same. The difference of 11% is probably because of the geometry. Geometry proportion does not meet the criterion for the experimental data in the Stress Concentration Factor diagram, [Eq. ③]. The Geometry proportion coefficient should be greater than 28.

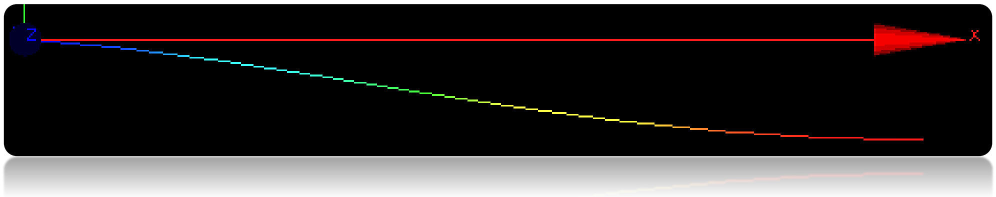
Redefining Mesh

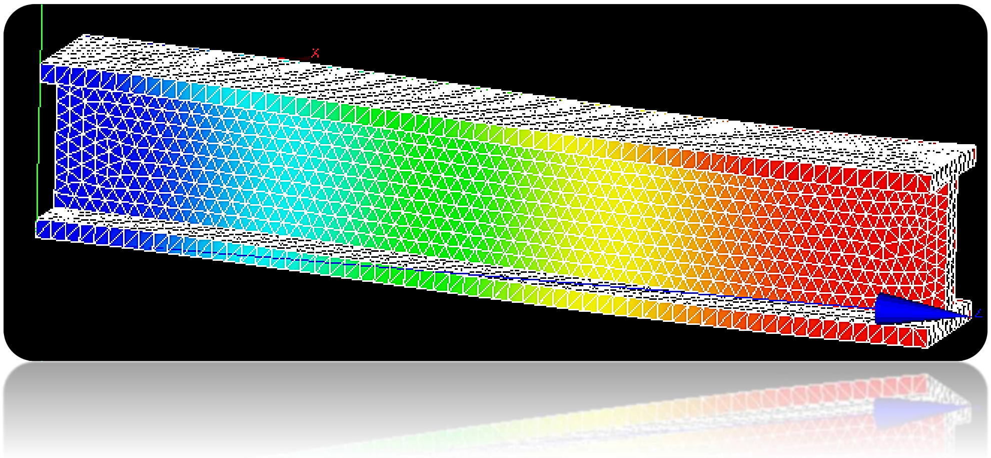
# We can see from the Post-Pro Module information results, that our mesh is a little irregular at some points, that is a very good reason to refine Mesh to almost half of its current size, and see what happens to our results.

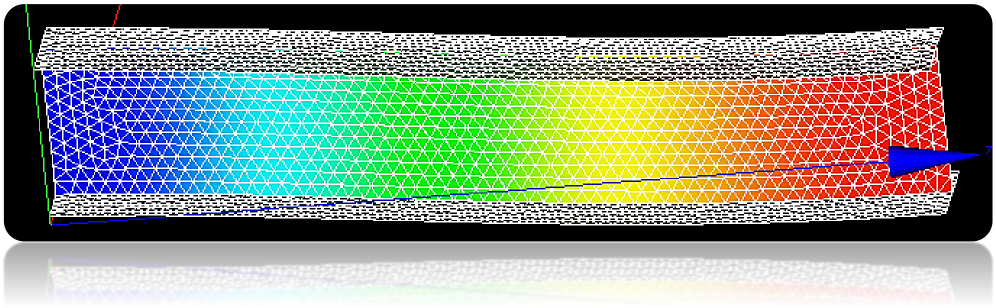
#	**Description**	**Figure**
1.	In order to do so, we need to activate Mesh Module again. After activating Mesh Module, open Mesh branch, open Hypotheses branch and right click to NETGEN 2D Parameters and reduce Max. and Min. Size to half of the current value. Do the same for Max Size_1 Hypothesis, and reduce to half the Length value. Right click on Mesh and Re-Compute it. Right click on Mesh and Convert to/from quadratic. Right click on Mesh and Re-Compute it.	 Hypothesis Construction Netgen 2D Arguments Local sizes OK Cancel HelpMax. Size 2 Min. Size 1 ☐ Second Order Fineness Moderate Growth Rate 0.3 Nb. Segs per Edge 1 Nb. Segs per Radius 2 ☐ Allow Quadrangles ☑ Optimize
2.	Next step is activating the Aster Module, in order to re-calculate the new adjustments. Right click to “linear-static” and choose Update Mesh. Right click to “linear-static” and choose Run.	
3.	Post-Pro branch is now visible. Activate Post-Pro Module from the drop down list. Open the branches until u reach RESU___SIGM_NOEU’s “0,-“ and right click on it. Choose Scalar Map with Scalar Mode = SIYY. Right click on it and choose Show Only.	
4.	The figure on the right, indicates my own results. We can see that the difference between these results and the previous ones is really small, but it is obvious that when we you change Mesh Parameters the results are changing also. In this case the difference is a decrease of 0.05%. A decrease like that, can hardly change the Stress Concentration Factor. $K_t = {}^{84.3428}/_{50} \; => \; K_t \approx 1.69$	

Chapter_VI: Level of Geometry

In this Chapter we are interested in comparing 1D, 2D, 3D
Analysis methods.
Beams, Solids and Shell Element Method.







Step 1: Purpose of the FE Analysis/ Description of the Problem

Reference --> *Autodesk Network Article.*

# **Beam Element -** A beam element is a slender structural member that offers resistance to forces and bending under applied loads. A beam element differs from a truss element in that a beam resists moments (twisting and bending) at the connections.

# These three node elements are formulated in three-dimensional space. The first two nodes (I-node and J-node) are specified by the element geometry. The third node (K-node) is used to orient each beam element in 3-D space (see Figure 1). A maximum of three translational degrees-of-freedom and three rotational degrees-of-freedom are defined for beam elements (see Figure 2). Three orthogonal forces (one axial and two shear) and three orthogonal moments (one torsion and two bending) are calculated at each end of each element. Optionally, the maximum normal stresses produced by combined axial and bending loads are calculated. Uniform inertia loads in three directions, fixed-end forces, and intermediate loads are the basic element based loadings.

# The basic guidelines for when to use a beam element are:

- The length of the element is much greater than the width or depth.
- The element has constant cross-sectional properties.
- The element must be able to transfer moments.
- The element must be able to handle a load distributed across its length.

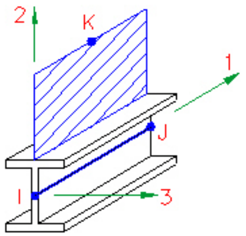


Figure 1: Beam Elements

# **Solid Element** – Solid elements are three-dimensional finite elements that can model solid bodies and structures without any a priori geometric simplification.

# Finite element models of this type have the advantage of directness. Geometric, constitutive and loading assumptions required to effect dimensionality reduction, for example to planar or axisymmetric behavior, are avoided. Boundary conditions on both forces and displacements can be more realistically treated. Another attractive feature is that the finite element mesh visually looks like the physical system.

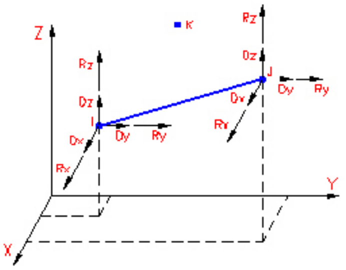


Figure 2: Beam Element Degrees-of-Freedom

# *Summarizing*, use of solid elements should be restricted to problem and analysis stages, such as verification, where the generality and flexibility of full 3D models is warranted. They should be avoided during design stages. Furthermore they should also be avoided in thin-wall structures such as aerospace shells, since solid elements tend to perform poorly because of locking problems.

# **Shell Element** – Shell elements are 4-8 node isoparametric quadrilaterals or 3-6 node triangular elements in any 3D orientation. The 4 node elements require a much finer mesh than the 8 node elements to give convergent displacements and stresses in models involving out-of-plane bending. Figure 1 shows some typical shell elements.

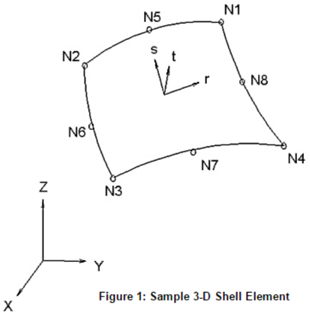


Figure 1: Sample 3-D Shell Element

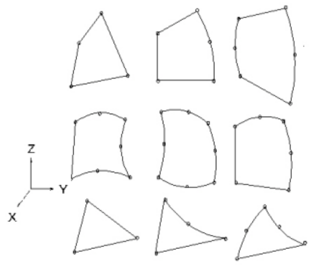


Figure 2: Typical Shell Elements

# The general and co-rotational shell element is formulated based on works by Ahmad, Iron and Zienkiewicz and later refined by Bathe and Balourchi. It can be applied to model both thick and thin shell problems. Also, the geometry of a doubly curved shell with variable thickness can be accurately described using the shell element.

# The thin shell element is based on thin plate theory. The bending behavior of the element is based on a discrete Kirchhoff approach to plate bending using Batoz's interpolation functions. This formulation satisfies Kirchhoff constraints along the boundary and provides linear variation of curvature through the element. The membrane behavior of the element is based on the Allman triangle which is derived from the Linear Strain Triangular (LST) element. A general curved surface is approximated by this element as a set of facets formed by the planes defined by the three nodes of each element. For these reason a well-refined mesh is necessary.

# The element geometry is described by the nodal point coordinates. Each shell element node has 5 degrees of freedom (DOF), three translations and two rotations. The translational DOF are in the global Cartesian coordinate system. The rotations are about two orthogonal axes on the shell surface defined at each node. The rotational boundary condition restraints and applied moments also refer to this nodal rotational system. The two rotational axes (V1 and V2) are usually automatically determined by the processor and you do not have to specifically orient them.

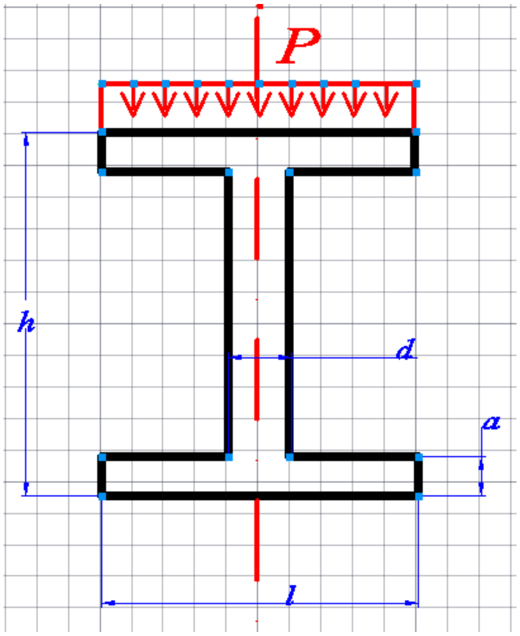
Step 2: Input Values for the FE Analysis

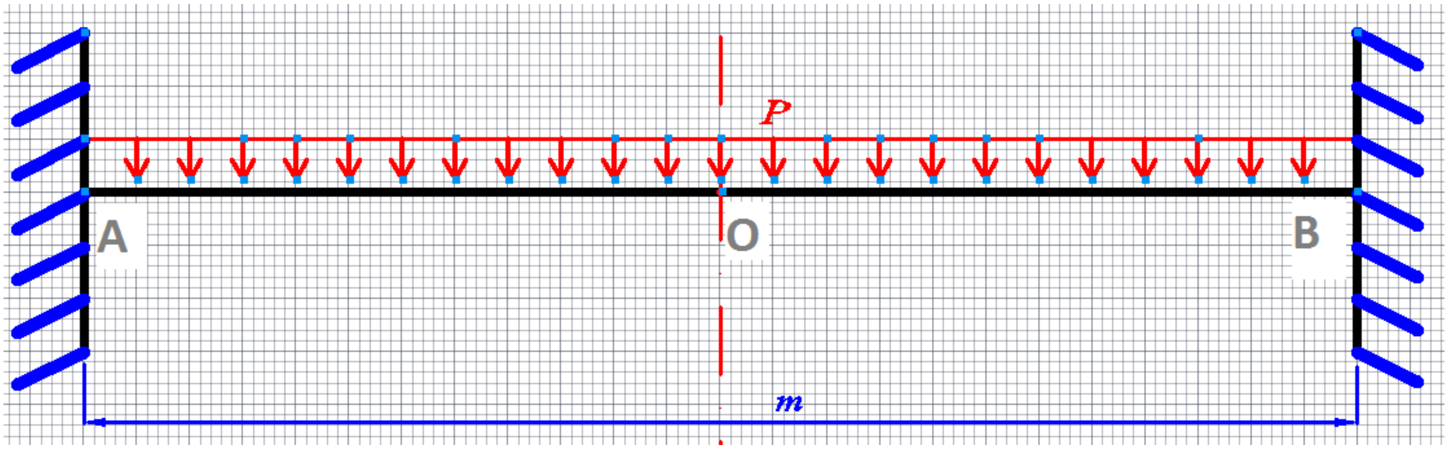
Chassis Dimensions:
Length = 115 mm [h];
Length = 100 mm [l];
Length = 20 mm [d];
Length = 12.5 mm [a];
Length = 1200 mm [m];

Young’s Modulus: 200 GPa;
Poisson’s Ratio: 0.3;
Pressure Applied: 10 MPa [P];

In case of Symmetry in “O”.

Supports: Fixed Support at “A” and “B” point.

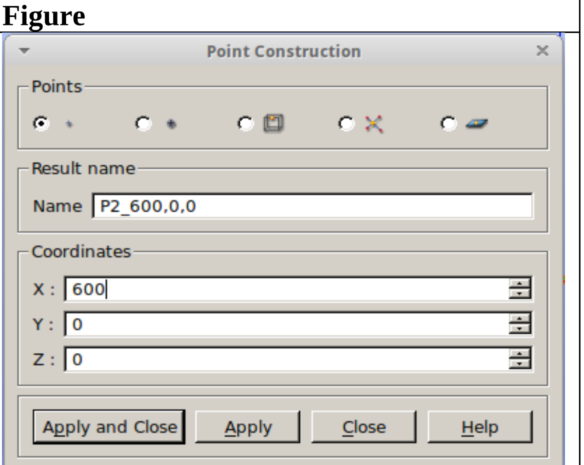
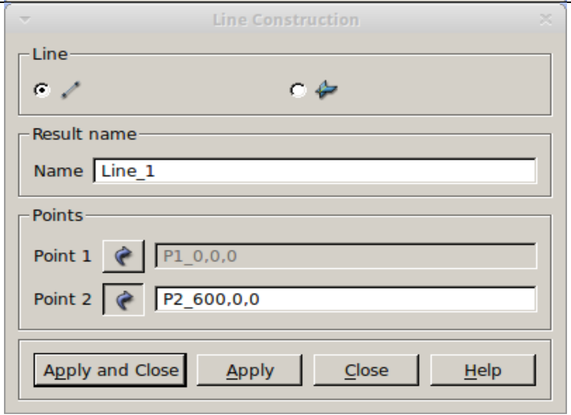
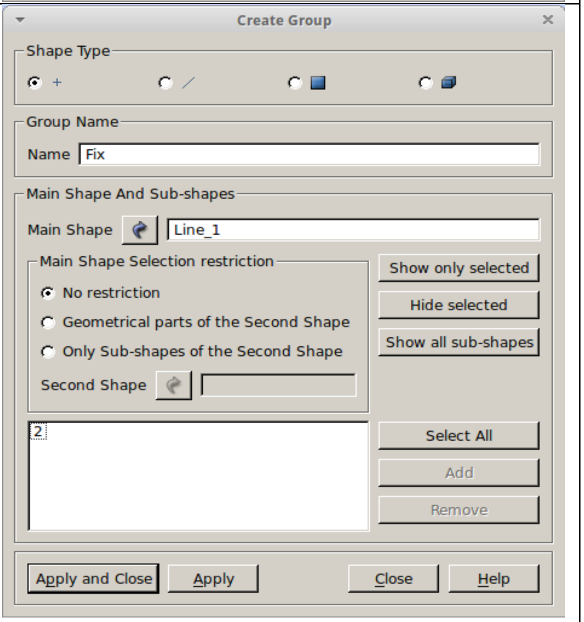




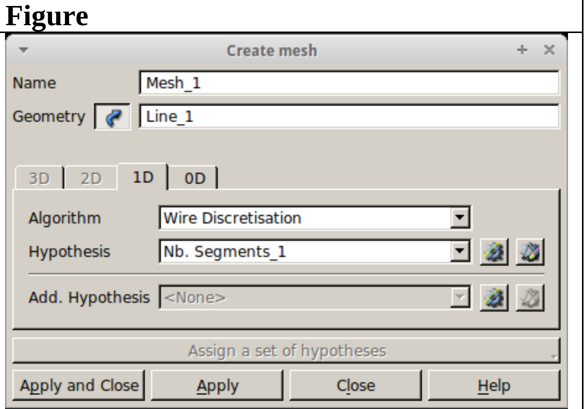
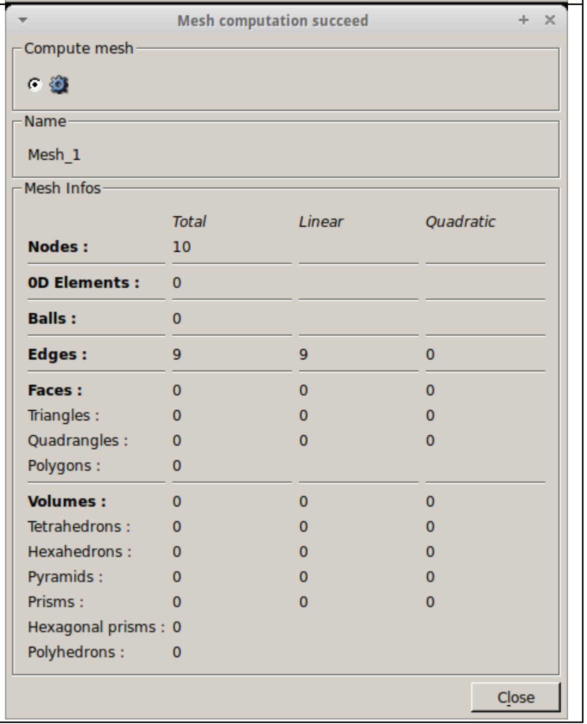
i. Beam_Elements

Step 3: Model Geometry

# Open up Salome_Meca, activate Geometry Module from the drop down list, save your study to a desired location and get ready to start modeling.

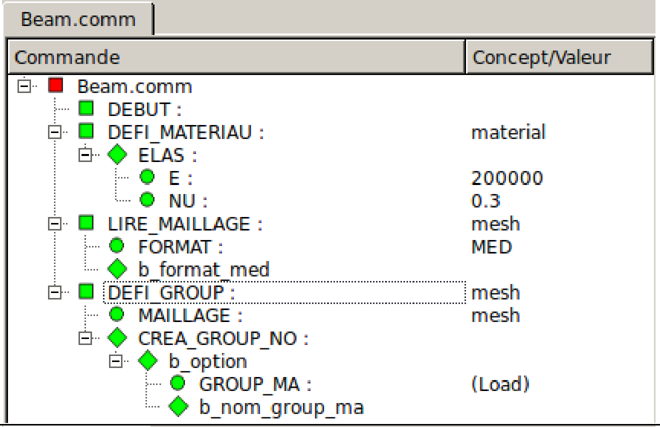
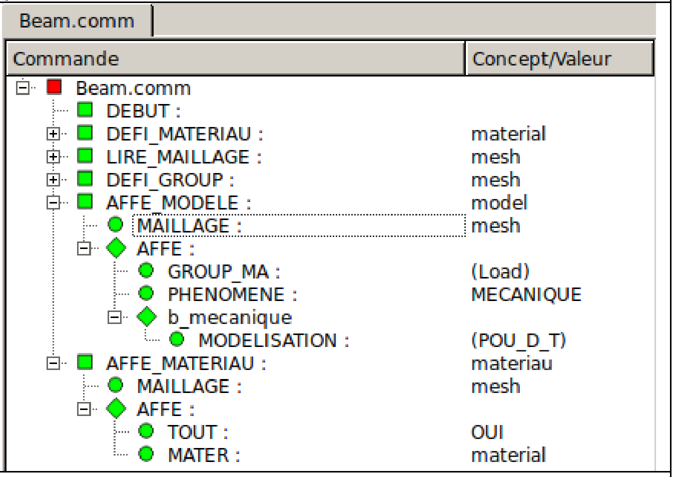
#	Description	Figure
1.	Creating our Beam should be an easy task. Go to New Entity → Basic → Point. Create two points in order to generate the line later. Point_1 = (0,0,0) Point_2 = (600,0,0) *Symmetric Model* Finish with "Apply and Close"	 Point Construction Name: P2_600,0,0 X : 600 Y : 0 Z : 0
2.	Next step, is creating our Line. Go to New Entity → Basic → Line. Insert P1_0,0,0 for the starting point. P2_600,0,0 for the end point. Finish with "Apply and Close"	 Line Construction Name: Line_1 Point 1: P1_0,0,0 Point 2: P2_600,0,0
3.	Final step to create our Geometry, would be adjusting our groups. Go to New Entity → Groups → Create. In the pop up window, select 1st Algorithm for Shape Type of Nodes. Enter Fix as a name, for the Fixed Support point, and "Add" the Point_1. Click "Apply" Do the same for the other end point, where we want our Displacement Support to be, and name it "Disp". Click "Apply" Lastly, select the 2nd Algorithm for Shape Type of Lines, name the group "Load" and "Add" the only line we got available. Finish with "Apply and Close"	 Create Group Name: Fix Main Shape: Line_1 Main Shape Selection restriction: ● No restriction 2
4.	These steps conclude out Geometry creation session. Save your project and continue to Mesh creation.	

Step 4: Meshing Geometry

#	Description	Figure
1.	Activate Mesh Module from the drop down list. Go to Mesh → Create Mesh. Select Line_1 for Geometry. Select Wire Discretisation for Algorithm. Select Nb. Segments_1 for Hypothesis, with number of Segments “9” (it might seem strange, but we will explain why we chose 9). Finish with “Apply and Close”	 Create mesh Name: Mesh_1 Geometry: Line_1 3D 2D 1D 0D Algorithm: Wire Discretisation Hypothesis: Nb. Segments_1 Add. Hypothesis: <None> Assign a set of hypotheses Apply and Close Apply Close Help
2.	Right-click on Mesh_1 → Compute. If everything is done correctly, you will have the same results as the figure on the right. *Hypothesis: Nb. Segments → The ability of this hypothesis is to separate your geometry to as many parts (in our case Edges) as you define. We set the number of segments to 9, which means that our mesh is going to consist of 9 Edges and 10 Nodes. Click “*Close*”	 Mesh computation succeed Compute mesh Name: Mesh_1 Mesh Infos (Total / Linear / Quadratic) **Nodes :** 10 **0D Elements :** 0 **Balls :** 0 **Edges :** 9 / 9 / 0 **Faces :** 0 / 0 / 0 Triangles : 0 / 0 / 0 Quadrangles : 0 / 0 / 0 Polygons : 0 **Volumes :** 0 / 0 / 0 Tetrahedrons : 0 / 0 / 0 Hexahedrons : 0 / 0 / 0 Pyramids : 0 / 0 / 0 Prisms : 0 / 0 / 0 Hexagonal prisms : 0 Polyhedrons : 0 Close

Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis

# In order to create our command file, this time we will use Eficas MultiCatalogue, which can prevent us from making grammatical mistakes in Code_Aster.

#	Description	Figure
1.	Go to Eficas → Eficas MultiCatalogue. In the pop-up window, click on Create New or Nouveau. Choose ASTER → "Ok", STA11 → "Ok" and you have set up Eficas and it is ready for modification. Before starting editing, save the command file in the same location where the geometry/mesh/.hdf files are currently located, with an appropriate name. Fishier → Enregistrer sous. We will explain step by step the commands we are going to use to complete our command file, because it is going to be a little complicated.	
2.	Beam.comm → Command file name. DEBUT → Start. DEFI_MATERIAU → Define Material. ELAS → Elastic (Constitutive law). E → Young's Modulus. NU → Poisson's Ratio. LIRE_MAILLAGE → Reading the mesh. MAILLAGE → Mesh. CREA_GROUP_NO → Create Nodal group. GROUP_MA → Group of elements.	
3.	AFFE_MODELE → Assign to the model. PHENOMENE → Phenomenon. MODELISATION → Modelization. AFFE_MATERIAU → Assign the material. TOUT → All.	
4.	Expanations: After starting the command file with the DEBUT command, we defined the material giving the material characteristics (Young's modulus and Poisson's Ratio). Then we specified the format of the mesh (MED). Next command defines where the mesh is going to be applied (in the group of elements named "Load"). Next figure starts with AFFE_MODELE, in this command we assign to the model, the kind of phenomenon we want to use, which will be mechanical and the type of modelization, POU_D_T (which is used for the Timoshenko beam theory). AFFE_MATERIAU assigns the mesh to the whole (keyword TOUT) material.	

```
Beam.comm
Commande                                  Concept/Valeur
Beam.comm
    DEBUT :
    DEFI_MATERIAU :                       material
        ELAS :
            E :                           200000
            NU :                          0.3
    LIRE_MAILLAGE :                       mesh
        FORMAT :                          MED
        b_format_med
    DEFI_GROUP :                          mesh
        MAILLAGE :                        mesh
        CREA_GROUP_NO :
            b_option
                GROUP_MA :                (Load)
                b_nom_group_ma
```

```
Beam.comm
Commande                                  Concept/Valeur
Beam.comm
    DEBUT :
    DEFI_MATERIAU :                       material
    LIRE_MAILLAGE :                       mesh
    DEFI_GROUP :                          mesh
    AFFE_MODELE :                         model
        MAILLAGE :                        mesh
        AFFE :
            GROUP_MA :                    (Load)
            PHENOMENE :                   MECANIQUE
            b_mecanique
                MODELISATION :            (POU_D_T)
    AFFE_MATERIAU :                       materiau
        MAILLAGE :                        mesh
        AFFE :
            TOUT :                        OUI
            MATER :                       material
```

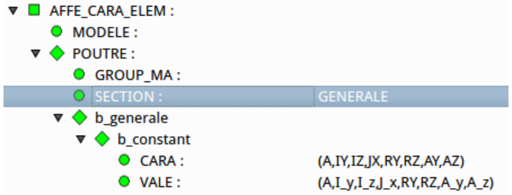
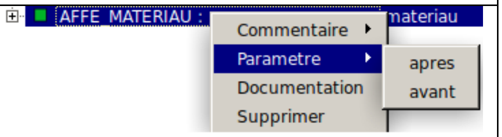
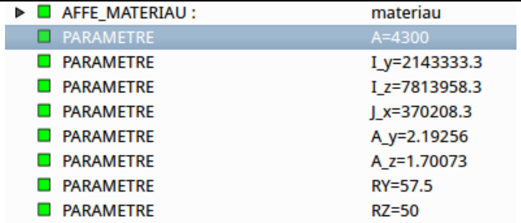
# Point of Reference: In the previous Beam System cases, after AFFE_MATERIAU, we used to assign the Cross Section to the Beam.

# In this case, we are asked to use I Cross-Section.

# Unfortunately, Code_Aster does not have a command that will give us the possibility to automatically adjust the I Section to the beam (C_A has Rectangular and Circular Cross Sections only).

# Code_Aster has Rectangular, Circular Cross Section options.

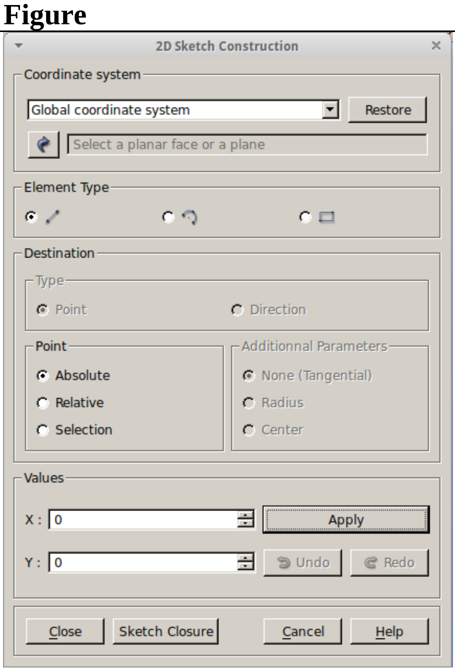
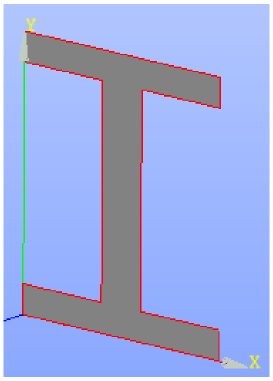
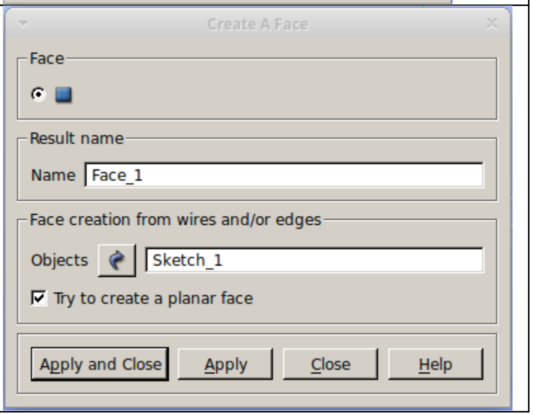
# But it also has an option called GENERALE, which gives you the ability to manually create whatever type of cross section you need.

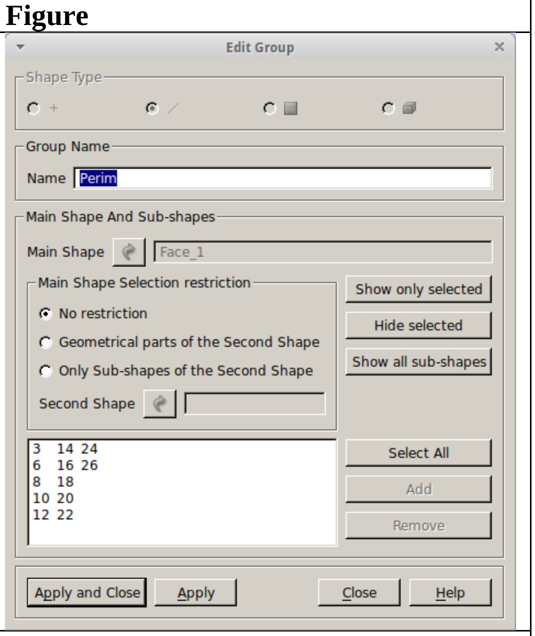
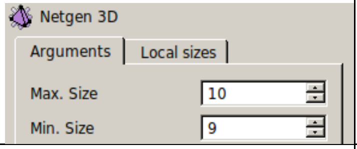
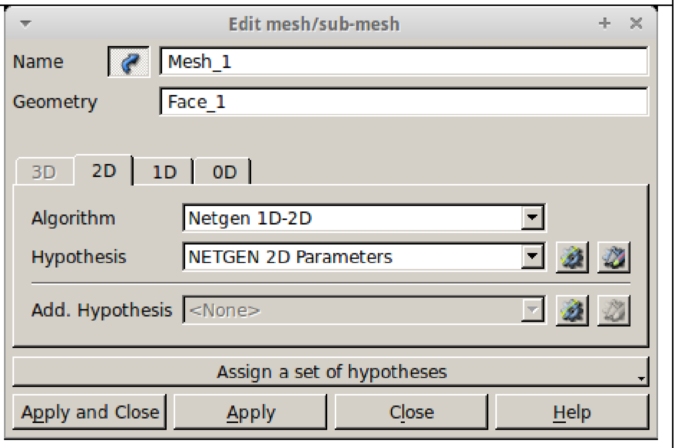
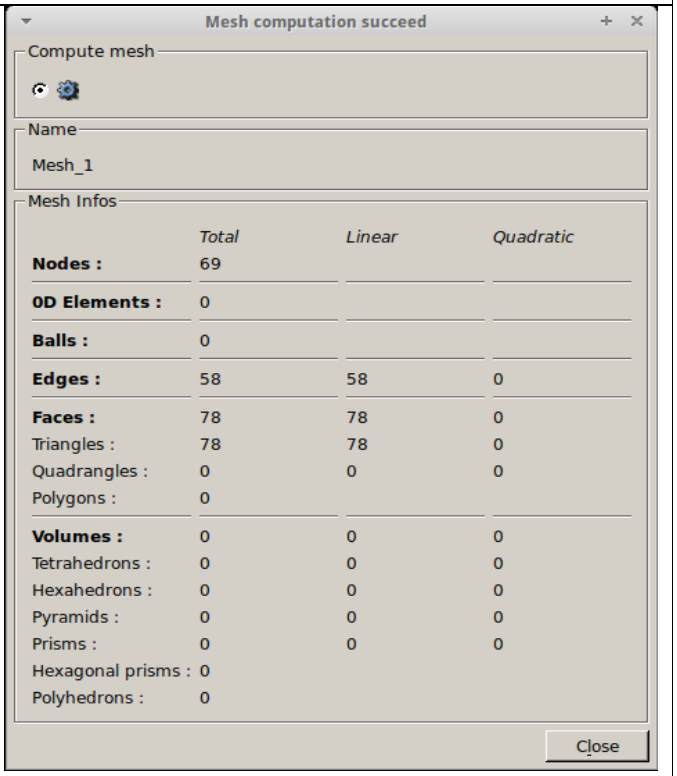
#	Description	Figure
1.	AFFE_CARA_ELEM is the command we use to assign our Cross Section to the Beam. SECTION: GENERALE, is the demanded point. GENERALE option though, needs some parameters in order to assign the correct cross section. The parameters that are needed for our Timoshenko beam (therefore "POU_D_T") are the following: A → Area. IY → Principal Geometrical Moment of Inertia compared to GZ. IZ → Principal Geometrical Moment of Inertia compared to GY. JX → Torsion Constant. RY → Distance from furthest left point of beam section to the centroid. RZ → Distance from bottom of beam section to the centroid. AY → Shear Coefficient of GY. AZ → Shear Coefficient of GZ.	 AFFE_CARA_ELEM : MODELE : POUTRE : GROUP_MA : SECTION : GENERALE b_generale b_constant CARA : (A,IY,IZ,JX,RY,RZ,AY,AZ) VALE : (A,I_y,I_z,J_x,RY,RZ,A_y,A_z)
2.	I will show you now, how to add Parameters in Eficas MultiCatalogue, and afterwards we will see how what we have to do, to acquire those values. If you right click on the last command you used, AFFE_MATERIAU in this case, you will see a pop up window like the one shown in the figure on the right. Go to Parametre → apres (=after). This means that Eficas will add a parameter, right after the AFFE_MATERIAU command.	 AFFE_MATERIAU : materiau Commentaire Parametre Documentation Supprimer apres avant
3.	These are all the values for each one of the parameters that the I Cross Section requires.	 AFFE_MATERIAU : materiau PARAMETRE A=4300 PARAMETRE I_y=2143333.3 PARAMETRE I_z=7813958.3 PARAMETRE J_x=370208.3 PARAMETRE A_y=2.19256 PARAMETRE A_z=1.70073 PARAMETRE RY=57.5 PARAMETRE RZ=50

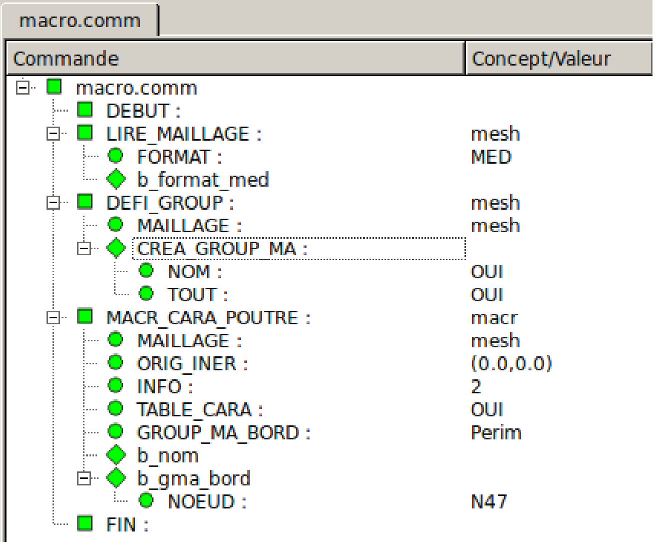
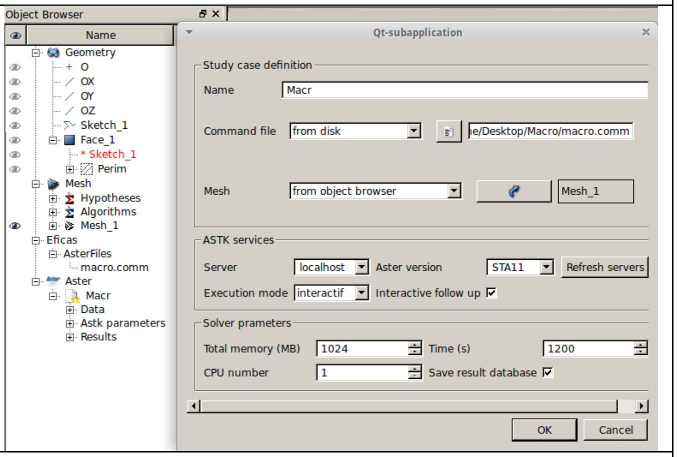
# Point: In order to acquire these values with Code_Aster, we will need to use a Macro command. In the next step we will show you how to do that.

Step $: Creating a Macro Command

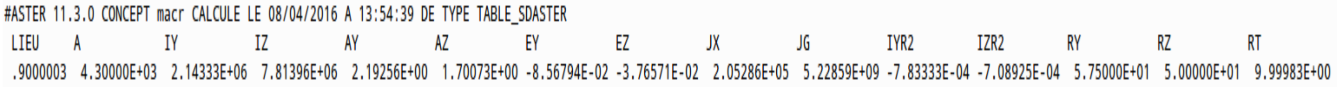
- # In order to acquire the parameters for the I Cross Section, we will need to create a macro command, as mentioned before.
- # The way this macro command works is the following:
  - We will make a sketch of a face for our I Section.
  - We will mesh that face.
  - We will create a command file, which after running the analysis will provide us (in the .resu file) all the necessary parameters.
- # Let's see that in action, open another Salome_Meca study case, save the case in another folder, activate Geometry Module from the drop down list and sketch the I Section.

#	Description	Figure
1.	We need to create the face in XY Plane**.** Go to New Entity → Basic → 2D Sketch. Leave everything Default, Coordinate system, Element Type, Destination, Point. Input the following Values: P_1 = (0,0) P_2 = (100,0) P_3 = (100,12.5) P_4 = (60,12.5) P_5 = (60,102.5) P_6 = (100,102.5) P_7 = (100,115) P_8 = (0,115) P_9 = (0,102.5) P_10 = (40,102.5) P_11 = (40,12.5) P_12 = (0,12.5) *Finish with "Sketch Closure".*	 2D Sketch Construction Coordinate system: Global coordinate system Destination – Type: Point; Point: Absolute; Additionnal Parameters: None (Tangential) Values: X : 0, Y : 0
2.	Creating a Face out of our Sketch. Go to New Entity → Build → Face. Input Sketch_1 for "Objects".  *Finish with "Apply and Close".*	 Create A Face Result name – Name: Face_1 Face creation from wires and/or edges – Objects: Sketch_1 ☑ Try to create a planar face

#	Description	Figure
3.	Next and final step, will be to create our groups. For this macro, we are going to need only one group, which will include the lines of the perimeter. Go to New Entity → Groups → Create. Select the 3rd Algorithm for Shape Type of Lines and click on "Select All", since we don't have any other lines. *Finish with "Apply and Close".*	Edit Group Group Name: Name: Perim Main Shape: Face_1 Main Shape Selection restriction: ● No restriction 3 14 24 6 16 26 8 18 10 20 12 22
4.	These steps conclude our Geometry, save in a proper location and move to Mesh Module.	
5.	Open Mesh Module from the drop down list. Go to Mesh → Create Mesh. Select Face_1 for Geometry. Input Netgen 1D-2D for Algorithm. Input NETGEN 2D Parameters for Hypothesis. Max. Size = 10. Min. Size = 9. *Finish with "Apply and Close".* Netgen 3D Max. Size: 10 Min. Size: 9	Edit mesh/sub-mesh Name: Mesh_1 Geometry: Face_1 2D Algorithm: Netgen 1D-2D Hypothesis: NETGEN 2D Parameters Add. Hypothesis: <None>
6.	Right click on Mesh_1 from the Object Browser → Compute. Your Mesh information should be the same as shown in the figure on the right. *Click "Close".*	Mesh computation succeed Name: Mesh_1 Mesh Infos: Total / Linear / Quadratic **Nodes :** 69 **0D Elements :** 0 **Balls :** 0 **Edges :** 58 / 58 / 0 **Faces :** 78 / 78 / 0 Triangles : 78 / 78 / 0 Quadrangles : 0 / 0 / 0 Polygons : 0 **Volumes :** 0 / 0 / 0 Tetrahedrons : 0 / 0 / 0 Hexahedrons : 0 / 0 / 0 Pyramids : 0 / 0 / 0 Prisms : 0 / 0 / 0 Hexagonal prisms : 0 Polyhedrons : 0

#	Description	Figure
7.	The figure on the right shows the commands that conclude the macro command file. Starting with DEBUT, assigning MED format to the mesh, defining mesh to the whole geometry. MACR_CARA_POUTRE → Macro command file. This will create a table in the .resu file with our parameters. ORIG_INER → Original axis orientation. GROUP_MA_BORD → Define the borders of the geometry with a group. In the part "NOEUD" you must assign a node from your mesh model, it does not matter which node you will chose.	 macro.comm Commande — Concept/Valeur macro.comm DEBUT : LIRE_MAILLAGE : mesh FORMAT : MED b_format_med DEFI_GROUP : mesh MAILLAGE : mesh CREA_GROUP_MA : NOM : OUI TOUT : OUI MACR_CARA_POUTRE : macr MAILLAGE : mesh ORIG_INER : (0.0,0.0) INFO : 2 TABLE_CARA : OUI GROUP_MA_BORD : Perim b_nom b_gma_bord NOEUD : N47 FIN :
8.	Activate Aster Module from the drop down list. Go to Aster → Add Study Case. Input the following data, and run the analysis.	 Study case definition Name: Macr Command file: from disk — e/Desktop/Macro/macro.comm Mesh: from object browser — Mesh_1 ASTK services Server: localhost Aster version: STA11 Execution mode: interactif Interactive follow up ☑ Solver prameters Total memory (MB): 1024 Time (s): 1200 CPU number: 1 Save result database ☑
9.	After running the analysis you will see that nothing happens to Salome_Meca Object Browser. Don't get frustrated, go to the folder where the .hdf, .comm, .mess files are located, and track down .resu file. Open it, and you should be able to see the following results, as shown in the figure below. These are all the parameters Code_Aster produced for us regarding the I Cross Section. Apparently, you can see by yourself that we do not need to use all of them (depends on our MODELISATION type).	

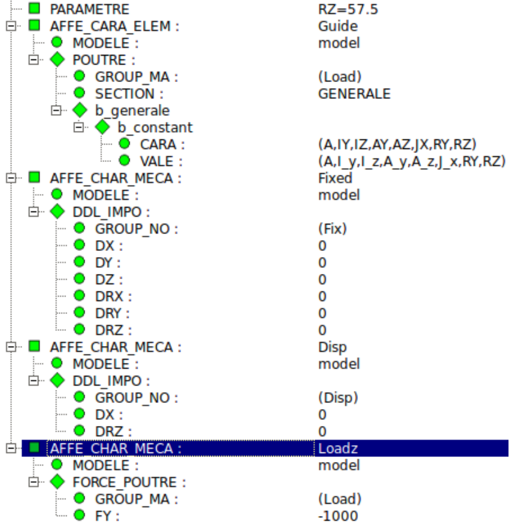
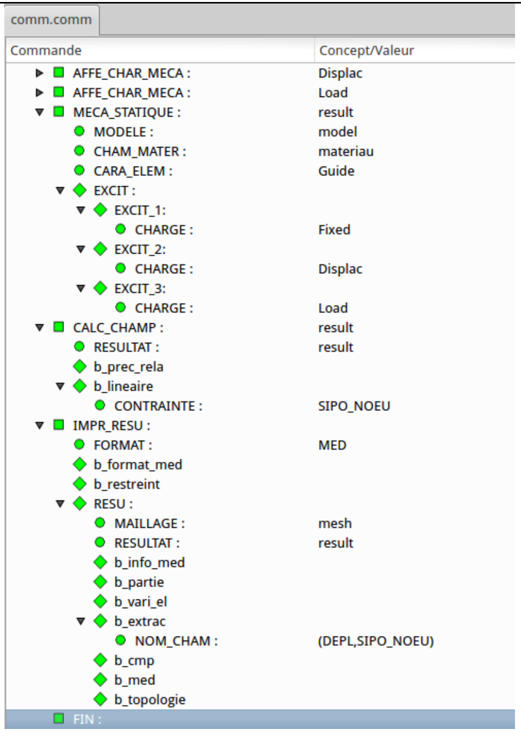
10.

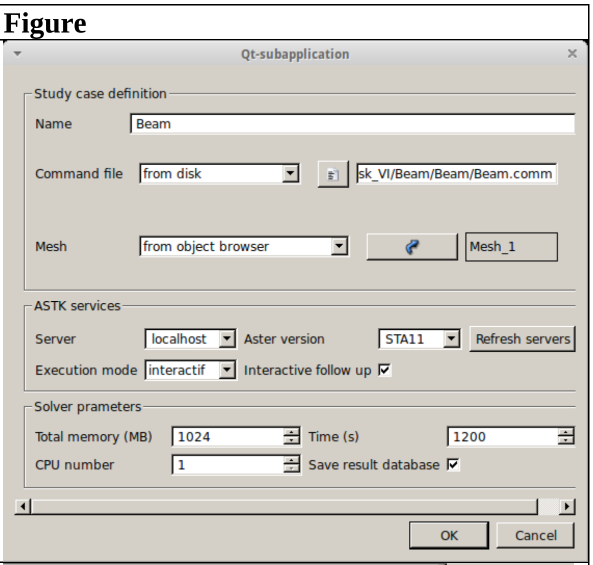
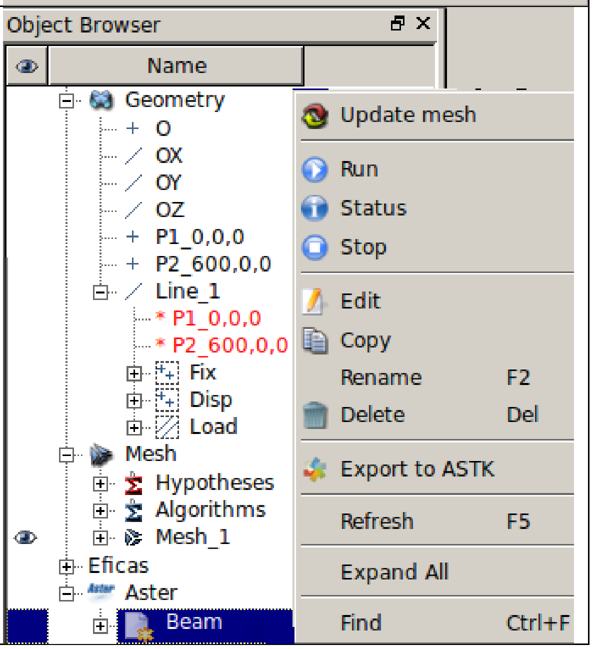


```
#ASTER 11.3.0 CONCEPT macr CALCULE LE 08/04/2016 A 13:54:39 DE TYPE TABLE_SDASTER
 LIEU     A           IY          IZ          AY          AZ          EY          EZ          JX          JG          IYR2        IZR2        RY          RZ          RT
 .9000003  4.30000E+03  2.14333E+06  7.81396E+06  2.19256E+00  1.70073E+00 -8.56794E-02 -3.76571E-02  2.05286E+05  5.22859E+09 -7.83333E-04 -7.08925E-04  5.75000E+01  5.00000E+01  9.99983E+00
```

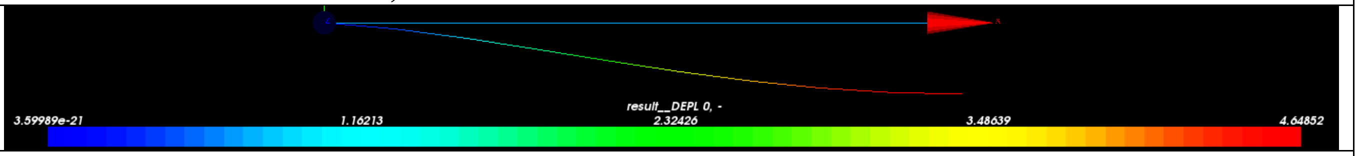
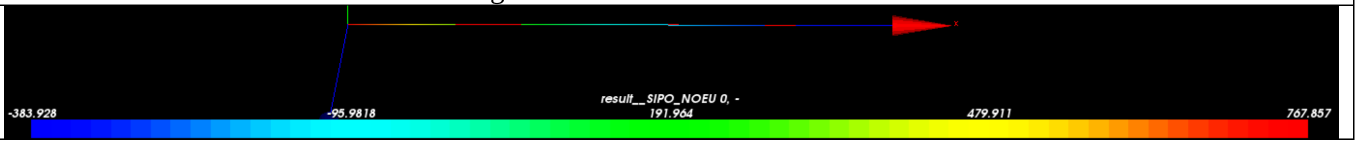
# This was the procedure we use to acquire the necessary parameters for the I Cross Section.

# Save this Macro Study and head back to our Beam Element case.

#	Description	Figure
4.	Now that we assigned the Parameters and also assigned the GENERALE Cross Section to our Beam, it is time for the Degress of Freedom and the Loads. AFFE_CHAR_MECA → Assigns characteristics to our model. Fixed Support to the group of Nodes named “Fix”. Displacement Support to the group of Nodes named “Disp”. Load of 1000 N/mm to the group of Elements named “Load”. # **Why 1.000 N/mm and not 10MPa [given data] ?** The equation for the Pressure equals with **P** = $F/_S$, where **F = Force** and **S = Area** of the surface where the Force is applied. **[F = P*S]** The equation for the Linear Pressure equals with $q = F/_l$, where **F = Force** and **l = Length** of the Beam. **[F = q*l]** In this study case we want the same Forces, consequently **P*S = q* l => q =** $P * S/_l$ ① ***S = h*w [h=height, w=width]*** ① → q = $10 * 60000/_{600}$ => $\boxed{q = 1000\ \text{N/mm}}$	 PARAMETRE RZ=57.5 AFFE_CARA_ELEM : Guide MODELE : model POUTRE : GROUP_MA : (Load) SECTION : GENERALE b_generale b_constant CARA : (A,IY,IZ,AY,AZ,JX,RY,RZ) VALE : (A,I_y,I_z,A_y,A_z,J_x,RY,RZ) AFFE_CHAR_MECA : Fixed MODELE : model DDL_IMPO : GROUP_NO : (Fix) DX : 0 DY : 0 DZ : 0 DRX : 0 DRY : 0 DRZ : 0 AFFE_CHAR_MECA : Disp MODELE : model DDL_IMPO : GROUP_NO : (Disp) DX : 0 DRZ : 0 AFFE_CHAR_MECA : Loadz MODELE : model FORCE_POUTRE : GROUP_MA : (Load) FY : -1000
5.	Next commands will be the followings: MECA_STATIQUE → Linear Static Mechanics with superposition of loads. CALC_CHAMP → Calculating the elements/nodes results. IMPR_RESU → Printing the results which we got from the CALC_CHAMP command.	 comm.comm Commande Concept/Valeur AFFE_CHAR_MECA : Displac AFFE_CHAR_MECA : Load MECA_STATIQUE : result MODELE : model CHAM_MATER : materiau CARA_ELEM : Guide EXCIT : EXCIT_1: CHARGE : Fixed EXCIT_2: CHARGE : Displac EXCIT_3: CHARGE : Load CALC_CHAMP : result RESULTAT : result b_prec_rela b_lineaire CONTRAINTE : SIPO_NOEU IMPR_RESU : FORMAT : MED b_format_med b_restreint RESU : MAILLAGE : mesh RESULTAT : result b_info_med b_partie b_vari_el b_extrac NOM_CHAM : (DEPL,SIPO_NOEU) b_cmp b_med b_topologie FIN :
6.	We are now done with the command file. Go to Fishier → Enregistrer, to save the configuration we made, and close Eficas MultiCatalogue.	

#	Description	Figure
	Activate Aster Module from the drop down list. Go to Aster → Add Study Case. Fill the gaps similar with the figure on your right. Locate your command file from your computer disk. Choose Mesh_1 as your Mesh. Change to STA11 for Aster Version Modify the Solver parameters. *Finish with "Ok".*	 Qt-subapplication Study case definition Name: Beam Command file: from disk, sk_VI/Beam/Beam/Beam.comm Mesh: from object browser, Mesh_1 ASTK services Server: localhost Aster version: STA11 Refresh servers Execution mode: interactif Interactive follow up ☑ Solver prameters Total memory (MB): 1024 Time (s): 1200 CPU number: 1 Save result database ☑ OK Cancel
	Your Object Browser should have the same looks as the figure on the right. Right click on Solids → Run. Your analysis will start running and when it finishes Post-Pro Branch should be visible. The analysis may take a while, since we used small size of elements and our solid model is a little bit bigger than the analysis we did so far.	 Object Browser Name Geometry O OX OY OZ P1_0,0,0 P2_600,0,0 Line_1 P1_0,0,0 P2_600,0,0 Fix Disp Load Mesh Hypotheses Algorithms Mesh_1 Eficas Aster Beam Update mesh Run Status Stop Edit Copy Rename F2 Delete Del Export to ASTK Refresh F5 Expand All Find Ctrl+F

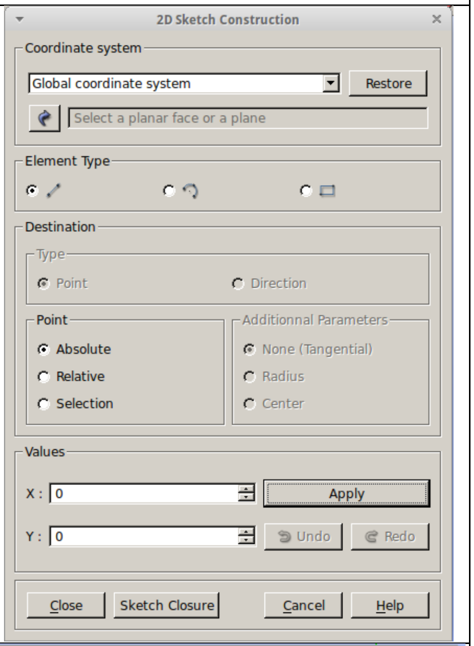
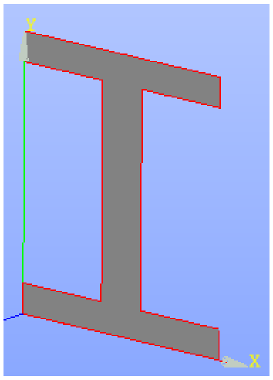
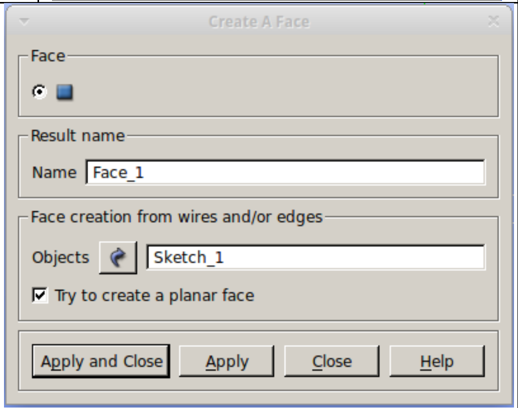
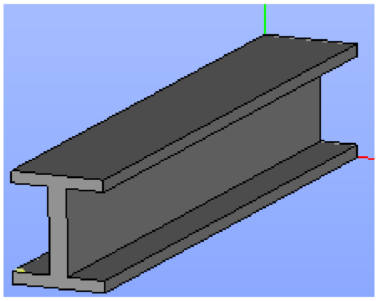
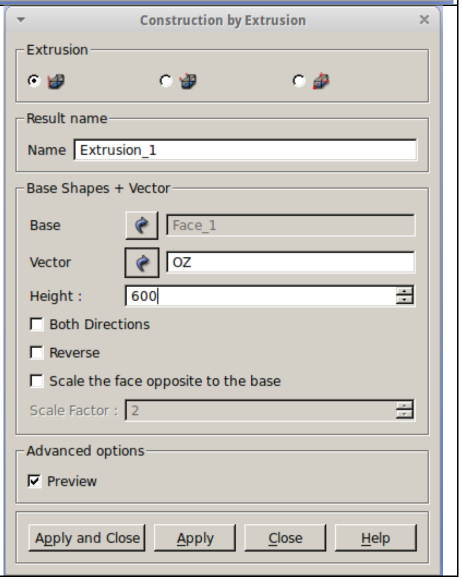
Step 11: Post Processing the Results

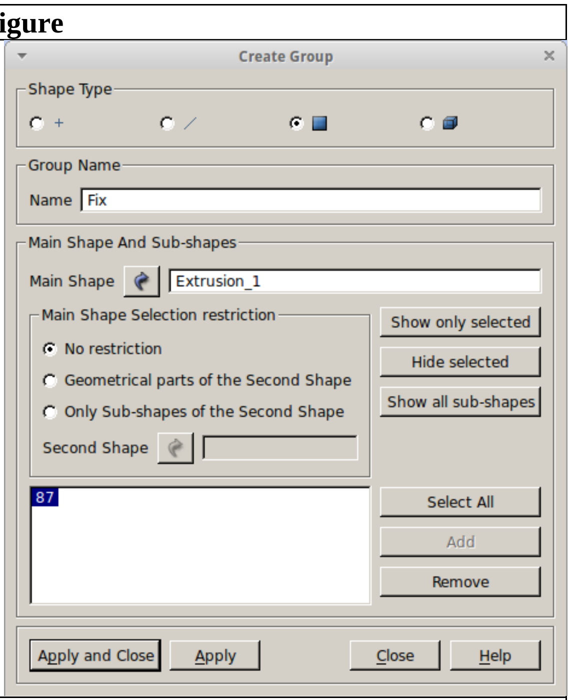
#	Description	Figure
1.	The steps to acquire the deformation and the stress result visualization are always the same, and you should be familiar with them until now. Right-click on 0,- → Deformed Shape → Tick Magnitude Coloring and change the Scale Factor to a bigger number, so you will be able to see the deformation more clear. Right-click on Def.Shape → Show Only → Representation → Surface. And you should have something similar to the figure below. Min. Deformation = 3.6e-21 mm, Max. Deformation = 4.65 mm	
2.	 3.59989e-21 1.16213 result__DEPL 0, - 2.32426 3.48639 4.64852	
3.	For the stress results: Result_SIPO_NOEU → Scalar Map → Mode: SMFZ (Stress due to bending Moment MZ) → Show Only. We can see that the direct stress has a range from -3.84 MPa to 767.9 MPa.	
4.	 -383.928 -95.9818 result__SIPO_NOEU 0, - 191.964 479.911 767.857	

ii. Solid_Elements

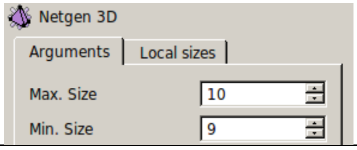
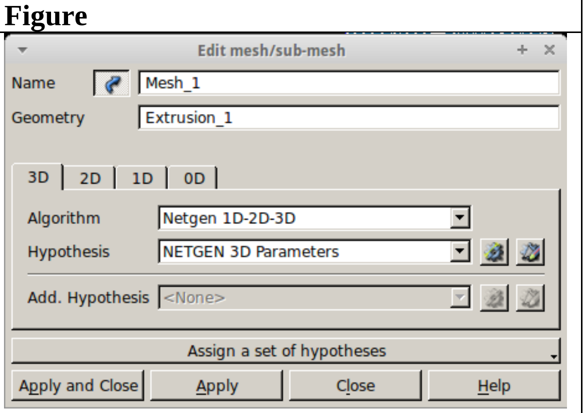
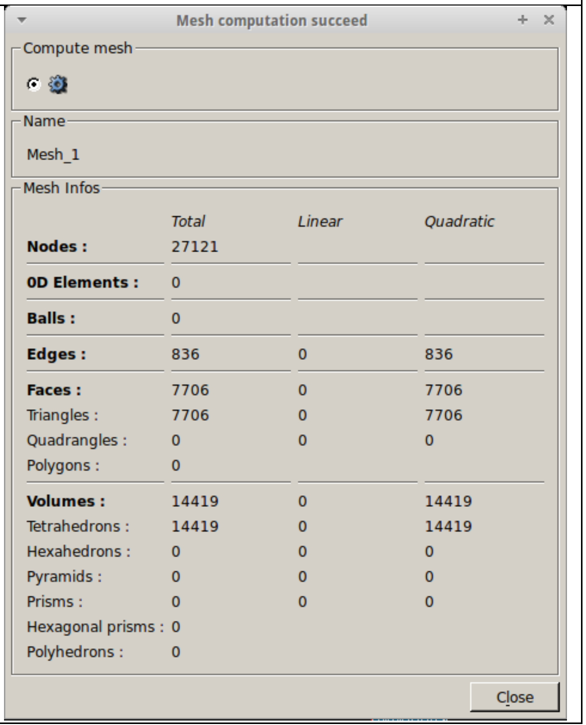
Step 3: Model Geometry

# Open up Salome_Meca, activate Geometry Module from the drop down list, save your study to a desired location and get ready to start modeling.

#	Description	Figure
1.	To create our Solid model, we will first create a face of our I Section and then we are going to Extrude it. Go to New Entity → Basic → 2D Sketch. Leave everything Default, Coordinate system, Element Type, Destination, Point. Input the following Values: P_1 = (0,0) P_2 = (100,0) P_3 = (100,12.5) P_4 = (60,12.5) P_5 = (60,102.5) P_6 = (100,102.5) P_7 = (100,115) P_8 = (0,115) P_9 = (0,102.5) P_10 = (40,102.5) P_11 = (40,12.5) P_12 = (0,12.5) *Finish with "Sketch Closure"*	 2D Sketch Construction Coordinate system: Global coordinate system Select a planar face or a plane Element Type Destination Type: ● Point ○ Direction Point: ● Absolute ○ Relative ○ Selection Additionnal Parameters: ● None (Tangential) ○ Radius ○ Center Values X : 0 Y : 0 Restore Apply Undo Redo Close Sketch Closure Cancel Help
2.	Creating a Face out of our Sketch. Go to New Entity → Build → Face. Input Sketch_1 for "Objects".  *Finish with "Apply and Close".*	 Create A Face Face Result name Name: Face_1 Face creation from wires and/or edges Objects: Sketch_1 ☑ Try to create a planar face
3.	Extruding our Face. Go to New Entity → Generation → Extrusion. Input the values according to the figure on the right.  *Finish with "Apply and Close".*	 Construction by Extrusion Extrusion Result name Name: Extrusion_1 Base Shapes + Vector Base: Face_1 Vector: OZ Height : 600 ☐ Both Directions ☐ Reverse ☐ Scale the face opposite to the base Scale Factor : 2 Advanced options ☑ Preview

#	Description	Figure
4.	Final step, will be to create our groups. Go to New Entity → Groups → Create. Select the 3^{rd} Algorithm for Shape Type of Faces and "Add" Fixed Support, Displacement Support, Loading and Guide group for the whole Solid Model (Select All). *Finish with "Apply and Close".*	 Create Group Shape Type: ○ ○ ● ○ Group Name Name: Fix Main Shape And Sub-shapes Main Shape: Extrusion_1 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape 87
5.	These steps conclude our Geometry, save in a proper location and move to Mesh Module.	

Step 4: Meshing Geometry

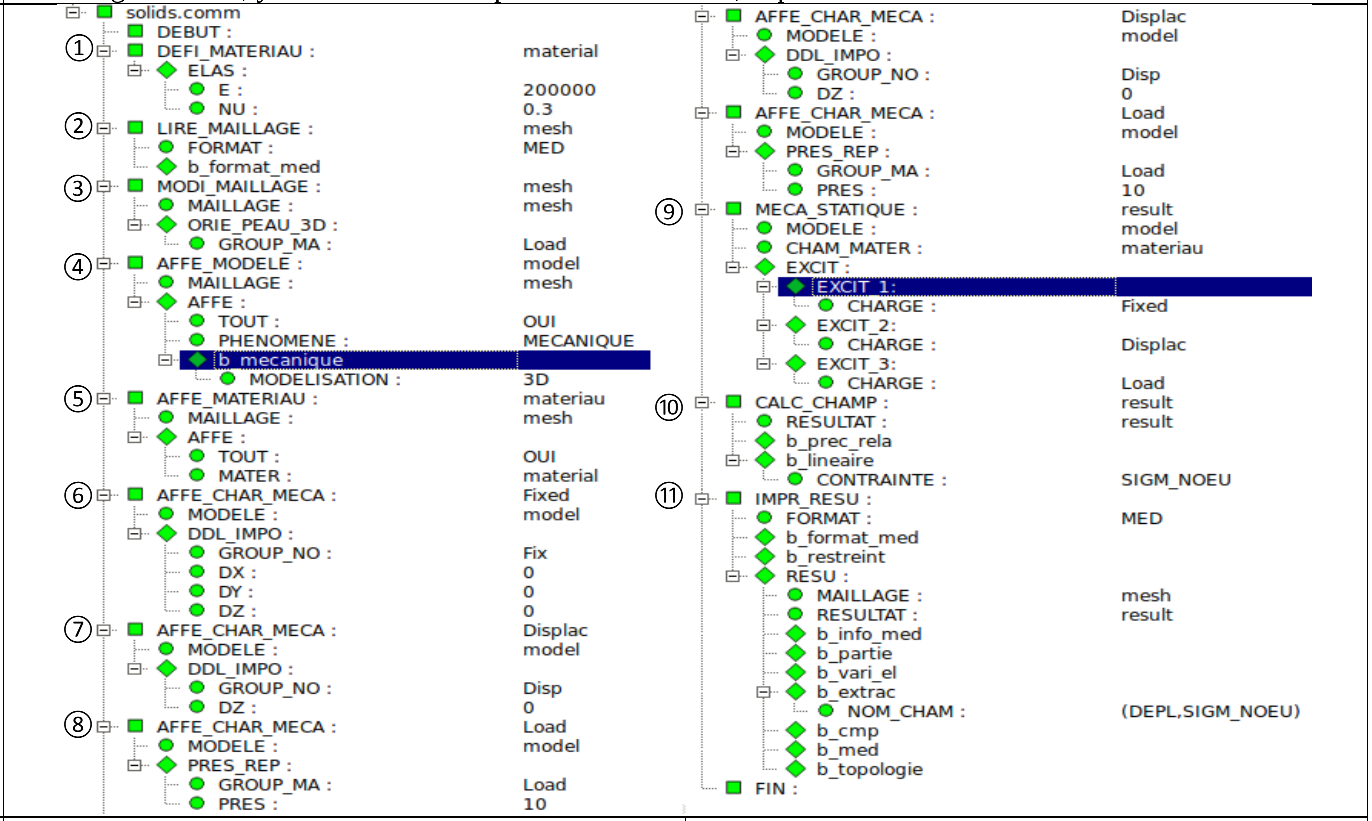
#	Description	Figure
1.	Open Mesh Module from the drop down list. Go to Mesh → Create Mesh. Select Extrusion_1 for Geometry. Input Netgen 1D-2D-3D for Algorithm. Input NETGEN 3D Parameters for Hypothesis. Max. Size = 10. Min. Size = 9.  Netgen 3D Max. Size: 10 Min. Size: 9 *Finish with “Apply and Close”.*	 Edit mesh/sub-mesh Name: Mesh_1 Geometry: Extrusion_1 Algorithm: Netgen 1D-2D-3D Hypothesis: NETGEN 3D Parameters Add. Hypothesis: <None>
2.	Right click on Mesh_1 from the Object Browser → Compute. Right click again on Mesh_1 → Convert to/from quadratic, to change our elements from Linear to Quadratic form. Right click again on Mesh_1 → Compute. Your Mesh information should be the same as shown in the figure on the right. *Click “Close”.*	 Mesh computation succeed Name: Mesh_1

Mesh Infos

	Total	*Linear*	*Quadratic*
Nodes :	27121		
0D Elements :	0		
Balls :	0		
Edges :	836	0	836
Faces :	7706	0	7706
Triangles :	7706	0	7706
Quadrangles :	0	0	0
Polygons :	0		
Volumes :	14419	0	14419
Tetrahedrons :	14419	0	14419
Hexahedrons :	0	0	0
Pyramids :	0	0	0
Prisms :	0	0	0
Hexagonal prisms :	0		
Polyhedrons :	0		

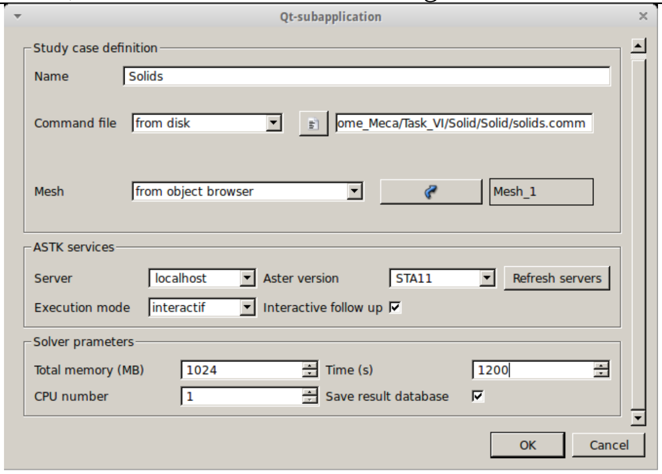
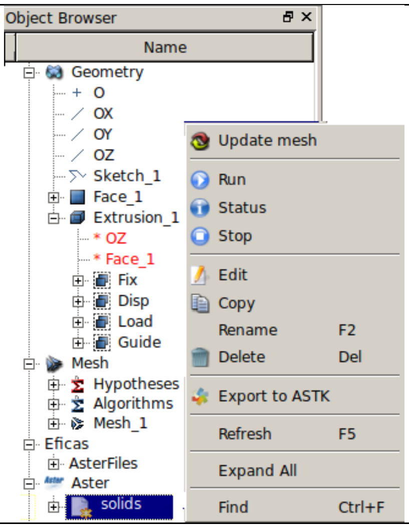
Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis

# Activate Eficas Module from the drop down list and start programming.

#	Description	Figure
1.	Go to Eficas → Eficas MultiCatalogue. In the pop-up window, click on Create New or Nouveau. Choose ASTER → "Ok", STA11 → "Ok" and you have set up Eficas and it is ready for modification. Before starting editing, save the command file in the same location where the geometry/mesh/.hdf files are currently located, with an appropriate name. Fishier → Enregistrer sous. In the figure below, you can see the completed command file, explanations will follow.	
2.		
3.	Starting with DEBUT command. 1. DEFI_MATERIAU → Defining the material, Structural Steel in this case. 2. LIRE_MAILLAGE → Reading the mesh, MED format. 3. MODI_MAILLAGE → Modifying the mesh, orientation of the normals of the skin of 3D elements. 4. AFFE_MODELE → Applying characteristics to the model, Mechanical phenomenon, and 3D modelization. 5. AFFE_MATERIAU → Applying the characteristics of the material to the mesh model.	6. AFFE_CHAR_MECA → Applying mechanical Supports, Fixed Support. 7. AFFE_CHAR_MECA → Applying mechanical Supports, Displacement Support. 8. AFFE_CHAR_MECA → Applying mechanical Loads, Pressure of 10 MPa. 9. MECA_STATIQUE → Linear Static Mechanics with superposition of supports and loads. 10. CALC_CHAMP → Calculating the elements/nodes results. 11. IMPR_RESU → Writing down, printing the results. Finish with FIN Command.

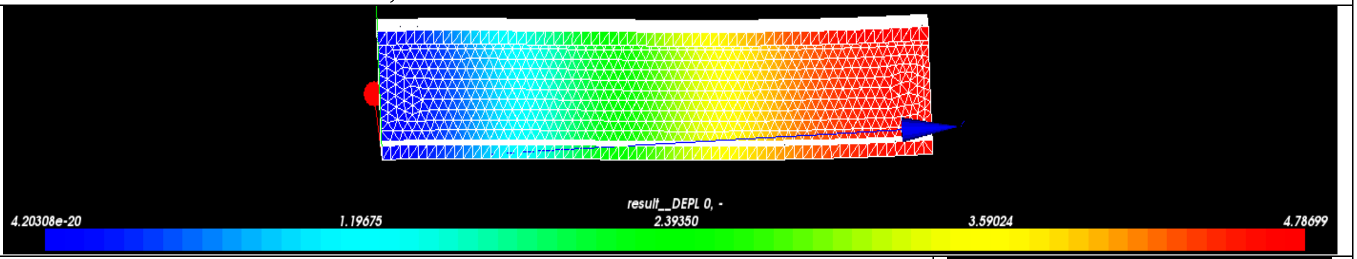
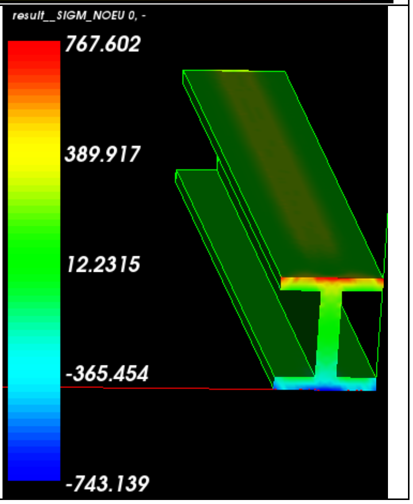
```
solids.comm
    DEBUT :
①   DEFI_MATERIAU :                 material
        ELAS :
            E :                     200000
            NU :                    0.3
②   LIRE_MAILLAGE :                 mesh
        FORMAT :                    MED
        b_format_med
③   MODI_MAILLAGE :                 mesh
        MAILLAGE :                  mesh
        ORIE_PEAU_3D :
            GROUP_MA :              Load
④   AFFE_MODELE :                   model
        MAILLAGE :                  mesh
        AFFE :
            TOUT :                  OUI
            PHENOMENE :             MECANIQUE
            b_mecanique
                MODELISATION :      3D
⑤   AFFE_MATERIAU :                 materiau
        MAILLAGE :                  mesh
        AFFE :
            TOUT :                  OUI
            MATER :                 material
⑥   AFFE_CHAR_MECA :                Fixed
        MODELE :                    model
        DDL_IMPO :
            GROUP_NO :              Fix
            DX :                    0
            DY :                    0
            DZ :                    0
⑦   AFFE_CHAR_MECA :                Displac
        MODELE :                    model
        DDL_IMPO :
            GROUP_NO :              Disp
            DZ :                    0
⑧   AFFE_CHAR_MECA :                Load
        MODELE :                    model
        PRES_REP :
            GROUP_MA :              Load
            PRES :                  10

    AFFE_CHAR_MECA :                Displac
        MODELE :                    model
        DDL_IMPO :
            GROUP_NO :              Disp
            DZ :                    0
    AFFE_CHAR_MECA :                Load
        MODELE :                    model
        PRES_REP :
            GROUP_MA :              Load
            PRES :                  10
⑨   MECA_STATIQUE :                 result
        MODELE :                    model
        CHAM_MATER :                materiau
        EXCIT :
            EXCIT_1:
                CHARGE :            Fixed
            EXCIT_2:
                CHARGE :            Displac
            EXCIT_3:
                CHARGE :            Load
⑩   CALC_CHAMP :                    result
        RESULTAT :                  result
        b_prec_rela
        b_lineaire
            CONTRAINTE :            SIGM_NOEU
⑪   IMPR_RESU :
        FORMAT :                    MED
        b_format_med
        b_restreint
        RESU :
            MAILLAGE :              mesh
            RESULTAT :              result
            b_info_med
            b_partie
            b_vari_el
            b_extrac
                NOM_CHAM :          (DEPL,SIGM_NOEU)
            b_cmp
            b_med
            b_topologie
    FIN :
```

# You will notice that in the section where we applied the supports, we didn't use rotation restrictions. That is because when it comes to solid elements there is no rotation. In Salome_Meca when you apply supports or loads, the modifications are applied to each node separately, so when you Fix a node in the top left part of the geometry and one more node in the bottom left part with DX, DY, DZ = 0 the model would not be able to rotate. That is why Rotation restrictions are redundant here.

#	Description	Figure
4.	We are now done with the command file. Go to Fishier → Enregistrer, to save the configuration we made, and close Eficas MultiCatalogue.	
5.	Activate Aster Module from the drop down list. Go to Aster → Add Study Case. Fill the gaps similar with the figure on your right. Locate your command file from your computer disk. Choose Mesh_1 as your Mesh. Change to STA11 for Aster Version Modify the Solver prameters. Finish with “Ok”	 Qt-subapplication Study case definition Name: Solids Command file: from disk; ome_Meca/Task_VI/Solid/Solid/solids.comm Mesh: from object browser; Mesh_1 ASTK services Server: localhost Aster version: STA11 Execution mode: interactif Interactive follow up ☑ Solver prameters Total memory (MB): 1024 Time (s): 1200 CPU number: 1 Save result database ☑
6.	Your Object Browser should have the same looks as the figure on the right. Right click on Solids → Run. Your analysis will start running and when it finishes Post-Pro Branch should be visible. The analysis may take a while, since we used small size of elements and our solid model is a little bit bigger than the analysis we did so far.	 Object Browser Name Geometry O OX OY OZ Sketch_1 Face_1 Extrusion_1 OZ Face_1 Fix Disp Load Guide Mesh Hypotheses Algorithms Mesh_1 Eficas AsterFiles Aster solids Update mesh Run Status Stop Edit Copy Rename F2 Delete Del Export to ASTK Refresh F5 Expand All Find Ctrl+F

Step 11: Post Processing the Results

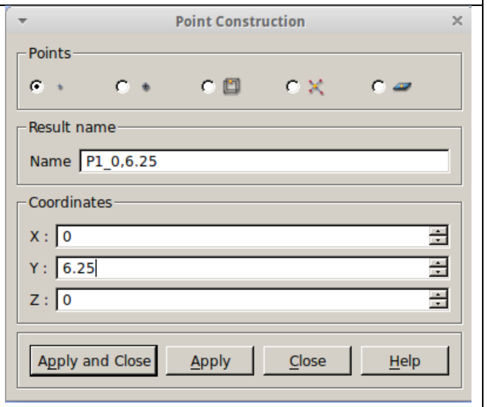
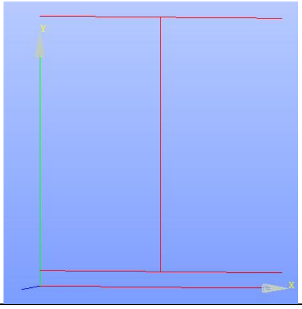
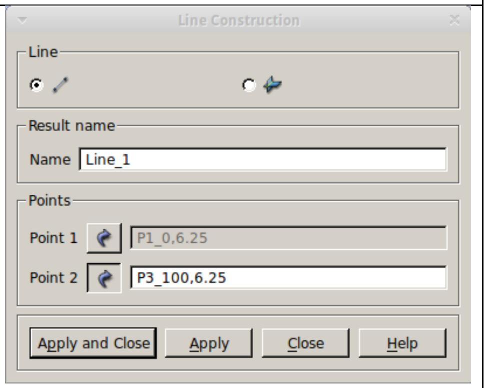
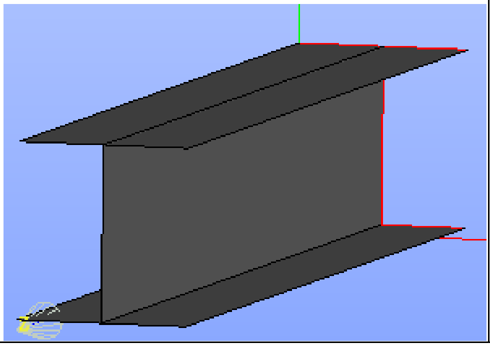
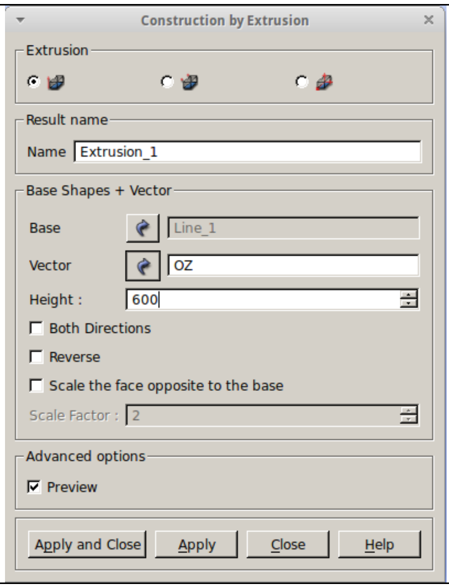
# Activate Post-Pro Module and start interrogating the results of our Finite Element Analysis.

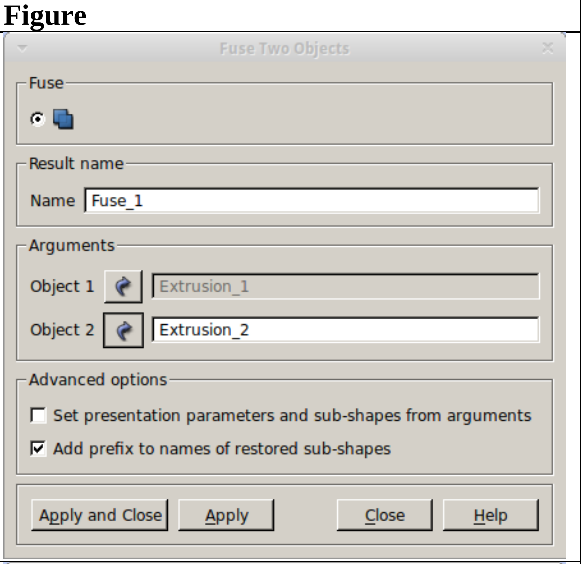
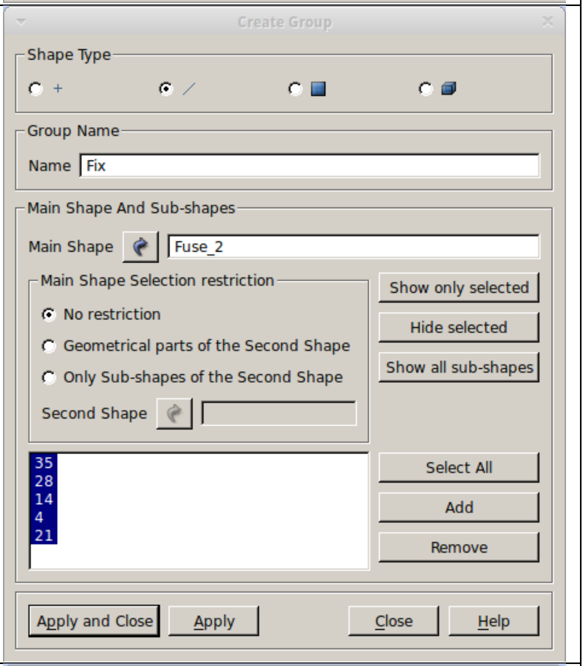
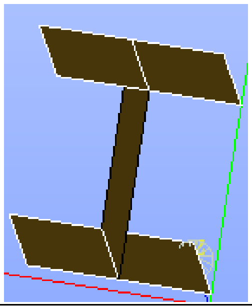
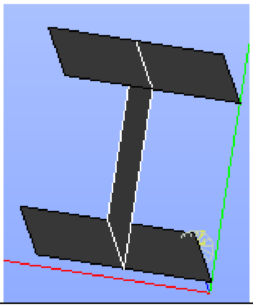
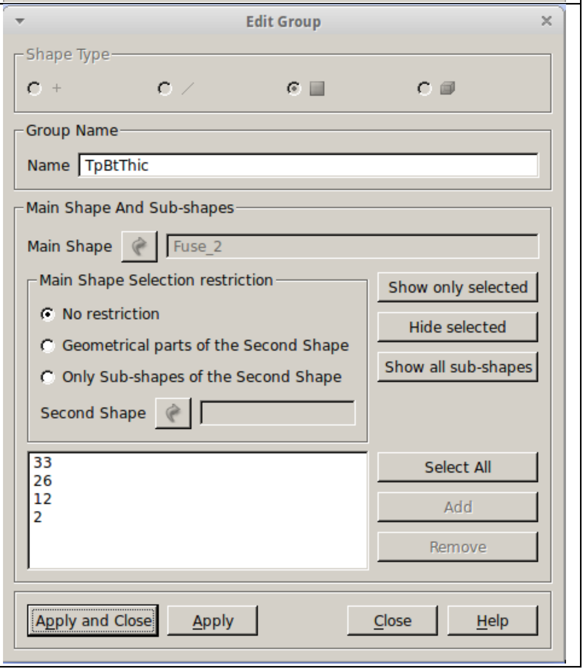
#	Description	Figure
1.	The steps to acquire the deformation and the stress result visualization are always the same, and you should be familiar with them until now. Right-click on 0,- → Deformed Shape → Tick Magnitude Coloring and change the Scale Factor to a bigger number, so you will be able to see the deformation more clear. Right-click on Def.Shape → Show Only → Representation → Surfaceframe. And you should have something similar to the figure below. Min. Deformation = 4.2e-20 mm, Max. Deformation = 4.79 mm	
2.	 result__DEPL 0, - 4.20308e-20 1.19675 2.39350 3.59024 4.78699	
3.	Same steps for the Normal Stress results. Right-click on 0,- → Scalar Map → Change Scalar Mode to SIXX. Right-click on Scalar Map → Show Only → Representation → Surface. And you should have something similar to the figure on the right. Min. Normal Stress = -743.1 MPa Max. Normal Stress = 767.6 MPa	 result__SIGM_NOEU 0, - 767.602 389.917 12.2315 -365.454 -743.139

iii. Surface_Elements

Step 3: Model Geometry

- # Open up Salome_Meca, activate Geometry Module from the drop down list, save your study to a desired location and get ready to start modeling.
- # Unfortunately Salome_Meca does not have an automated option to acquire the mid-surfaces out of our solid model, like ANSYS Workbench or other software. We will have to sketch down the Surface model by ourselves.

#	Description	Figure
1.	In order to create our Shell model, first we will need to create the middle lines of every face, and then Extrude the lines. Go to New Entity → Basic → Point. Input the following points: P_1 = (0,6.25) P_2 = (50,6.25) P_3 = (100,6.25) P_4 = (0,108.75) P_5 = (50,108.75) P_6 = (100,108.75) *Finish with "Apply and Close".*	 Point Construction Result name Name: P1_0,6.25 Coordinates X : 0 Y : 6.25 Z : 0
2.	Next step is to connect the Points with Lines. Be careful here with the points because it can be a little complicated. Go to New Entity → Basic → Line. Connect the following Points: P_1 with P_3. Click "Apply" P_4 with P_6. Click "Apply" P_2 with P_5. *Finish with "Apply and Close".* 	 Line Construction Result name Name: Line_1 Points Point 1: P1_0,6.25 Point 2: P3_100,6.25
4.	Extruding the Lines. Go to New Entity → Generation → Extrusion. Fill in the gaps according to the figure on the right. And do the same for Line_2 and Line_3.  *Finish with "Apply and Close".*	 Construction by Extrusion Result name Name: Extrusion_1 Base Shapes + Vector Base: Line_1 Vector: OZ Height : 600 ☐ Both Directions ☐ Reverse ☐ Scale the face opposite to the base Scale Factor : 2 Advanced options ☑ Preview

#	Description	Figure
5.	After creating the 3 different extrusions from the lines, we need to somehow connect them. Fuse command will help us with that. Go to Operation → Boolean → Fuse. Here is the tricky part, Fuse commands gives allow us to use only 2 arguments to Fuse, and we need 3 (3 lines, 3 extrusions). To make it work, first we will chose 2 out of 3 extrusions, like the figure on the right. Finish with "Apply and Close" Go to Operation → Boolean → Fuse again, but this time for Object 1 input the "Fuse_1" we created before, and for Object_2 input the extrusion you didn't use before. Finish with "Apply and Close"	 Fuse Two Objects Fuse ● Result name Name Fuse_1 Arguments Object 1 Extrusion_1 Object 2 Extrusion_2 Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes Apply and Close Apply Close Help
6.	Last step is creating our groups. Go to New Entity → Group → Create. Select the 2nd Algorithm for Shape Type Lines. Fixed Support: Add the lines where we want our fixed support to be located. You will notice that there are 5 lines instead of 3, that would not be a problem. Displacement Support: Add the lines where we want our displacement support to be located, opposite side of fixed support. Select the 3rd Algorithm for Shape Type Faces. Loads: Add the top faces where we need to apply our pressure. Guide: Select All, to add everything in the Guide group.	 Create Group Shape Type ○ + ● ⁄ ○ ○ Group Name Name Fix Main Shape And Sub-shapes Main Shape Fuse_2 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape Show only selected Hide selected Show all sub-shapes 35 28 14 4 21 Select All Add Remove Apply and Close Apply Close Help
7.	In this study case, we are going to need two more groups, which will be describing the different thicknesses we got. Selecting the 3rd Algorithm again for Shape Type Faces. Group Name: TpBtThic (Top and Bottom Thickness) Add the top and bottom faces which have thickness of 12.5mm. Group Name: MidThic (Middle Thickness) Add the middle face which has thickness of 20mm.  	 Edit Group Shape Type ○ + ○ ⁄ ● ○ Group Name Name TpBtThic Main Shape And Sub-shapes Main Shape Fuse_2 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape Second Shape Show only selected Hide selected Show all sub-shapes 33 26 12 2 Select All Add Remove Apply and Close Apply Close Help

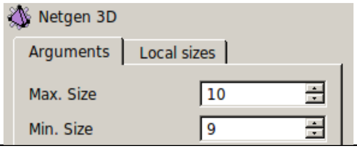
Step 4: Meshing Geometry

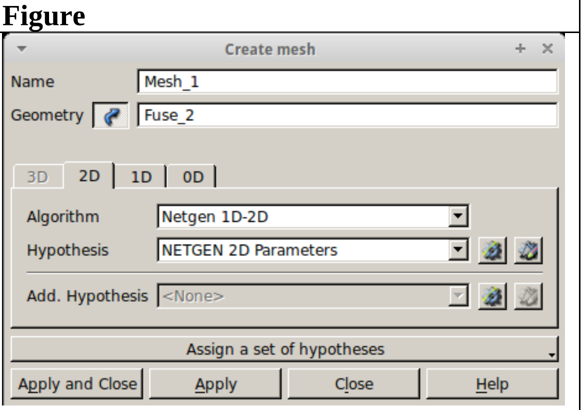
<table>
<tr><th>#</th><th>Description</th><th>Figure</th></tr>
<tr><td><i>1.</i></td><td>Open Mesh Module from the drop down list.

Go to Mesh → Create Mesh.

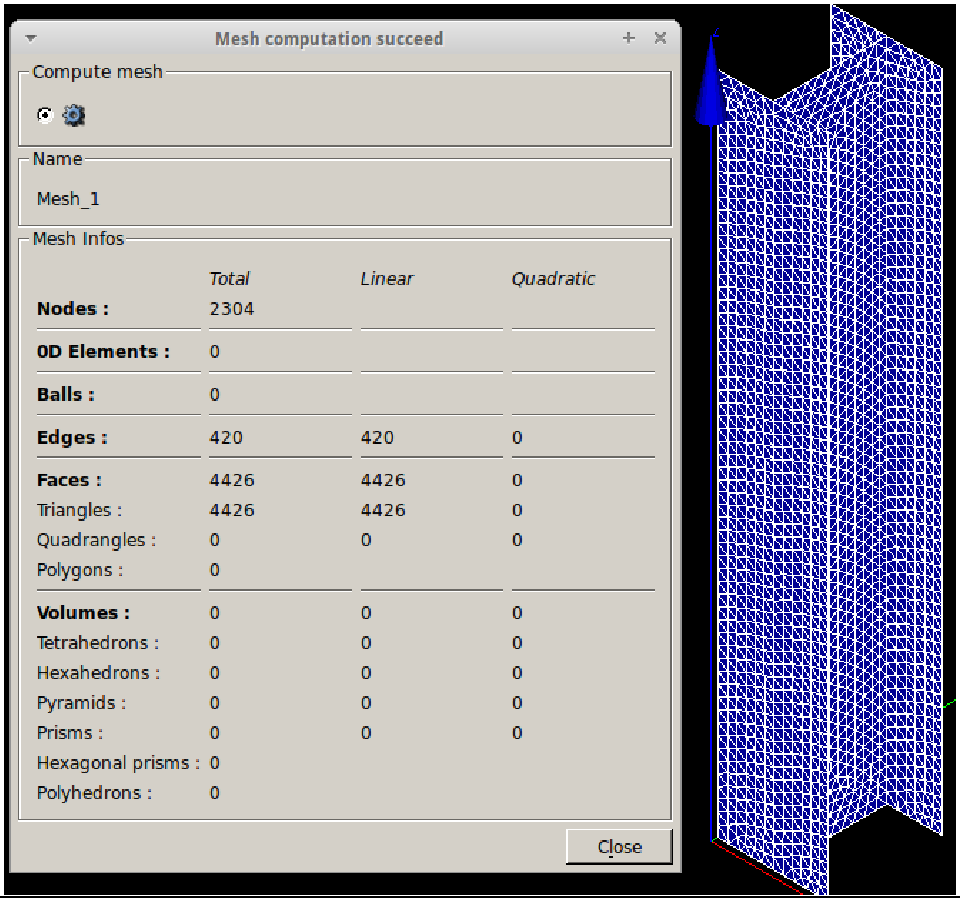
Select Fuse_2 for Geometry.

Input Netgen 1D-2D-3D for Algorithm.
Input NETGEN 3D Parameters for Hypothesis.
Max. Size = 10.
Min. Size = 9.

<i>Finish with “Apply and Close”.</i>


Max. Size: 10
Min. Size: 9</td><td>

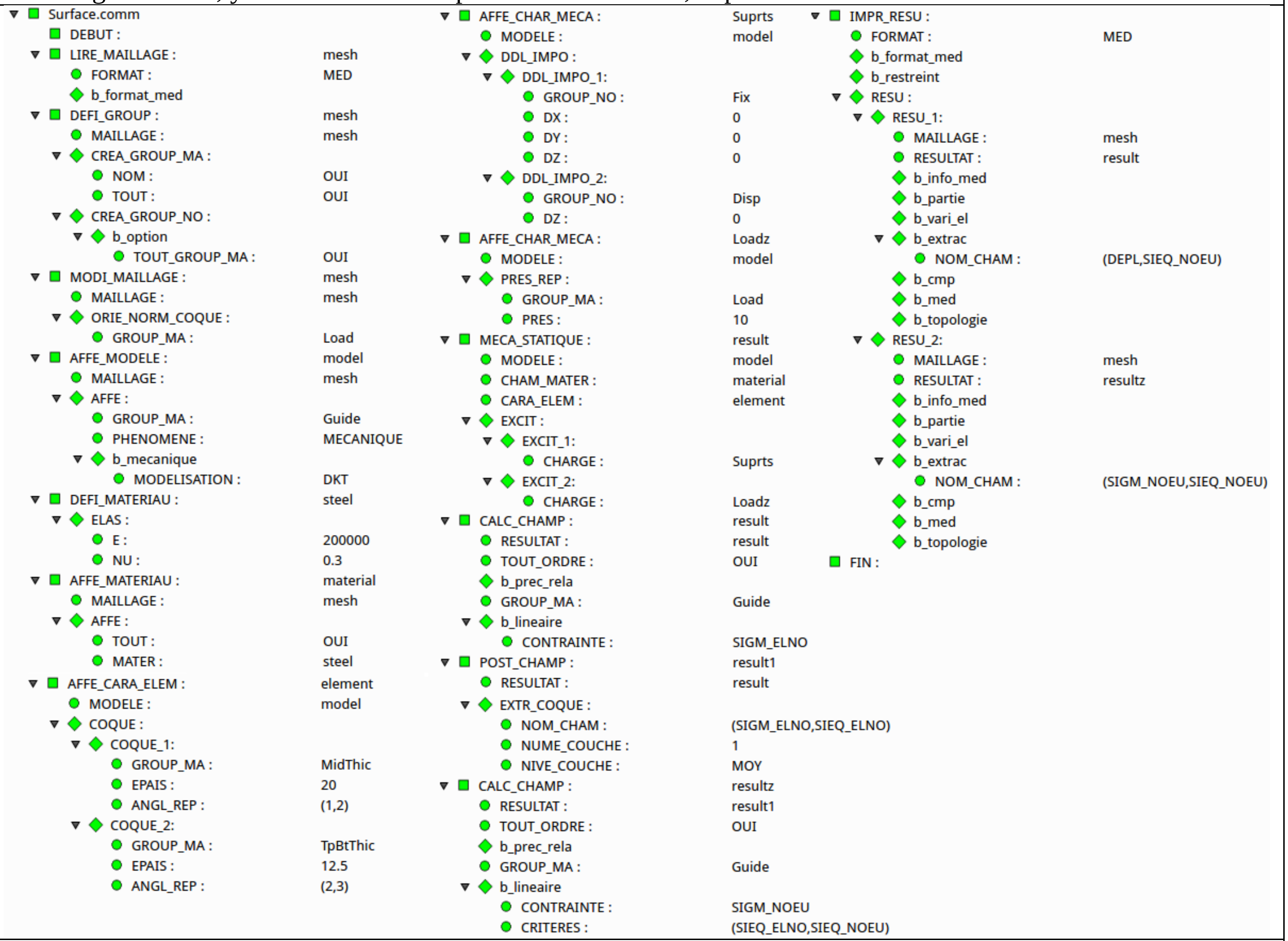
Create mesh
3D
2D
1D
0D
Assign a set of hypotheses
Apply and Close
Apply
Close
Help

Name: Mesh_1
Geometry: Fuse_2
Algorithm: Netgen 1D-2D
Hypothesis: NETGEN 2D Parameters
Add. Hypothesis: &lt;None&gt;</td></tr>
<tr><td><i>2.</i></td><td colspan="2">Right click on Mesh_1 from the Object Browser → Compute.


Name
Mesh_1
Mesh Infos
<table>
<tr><th></th><th><i>Total</i></th><th><i>Linear</i></th><th><i>Quadratic</i></th></tr>
<tr><td>Nodes :</td><td>2304</td><td></td><td></td></tr>
<tr><td>0D Elements :</td><td>0</td><td></td><td></td></tr>
<tr><td>Balls :</td><td>0</td><td></td><td></td></tr>
<tr><td>Edges :</td><td>420</td><td>420</td><td>0</td></tr>
<tr><td>Faces :</td><td>4426</td><td>4426</td><td>0</td></tr>
<tr><td>Triangles :</td><td>4426</td><td>4426</td><td>0</td></tr>
<tr><td>Quadrangles :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Polygons :</td><td>0</td><td></td><td></td></tr>
<tr><td>Volumes :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Tetrahedrons :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Hexahedrons :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Pyramids :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Prisms :</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>Hexagonal prisms :</td><td>0</td><td></td><td></td></tr>
<tr><td>Polyhedrons :</td><td>0</td><td></td><td></td></tr>
</table></td></tr>
</table>

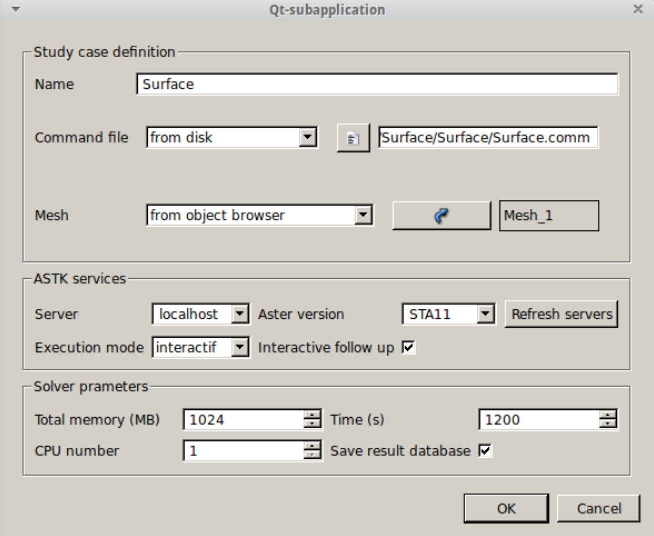
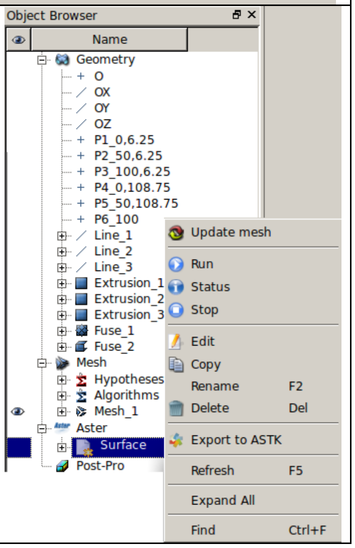
Step 5, 6, 7, 8, 9, 10: Creating the command file, run the analysis

# Activate Eficas Module from the drop down list and start programming.

#	Description	Figure
1.	Go to Eficas → Eficas MultiCatalogue. In the pop-up window, click on Create New or Nouveau. Choose ASTER → “Ok”, STA11 → “Ok” and you have set up Eficas and it is ready for modification. Before starting editing, save the command file in the same location where the geometry/mesh/.hdf files are currently located, with an appropriate name. Fishier → Enregistrer sous. In the figure below, you can see the completed command file, explanations will follow.	
2.		
3.	Up until now you must have acquired enough knowledge to be able to understand the purpose of each command. For that reason we will discuss only the newly added commands. # ORIE_NORM_COQUE (MODI_MAILAGE) → Orientation of the normals for the Shell Element. # COQUE_1/_2 (AFFE_CARA_ELEM) → Applying thickness to the given groups (we got two difference thicknesses in our geometry). # ANGLE_REP (AFFE_CARA_ELEM) → Nautic Angles, used to define a local coordinate in the tangent plane of a 2D element. # POST_CHAMP → In order to visualize Normal Stress results (SIGM_NOEU) in particular section of thickness (moy - bottom, sup - top), we have to include this command, because DKT Modelization only allows us to prin Equivalent Stress.	

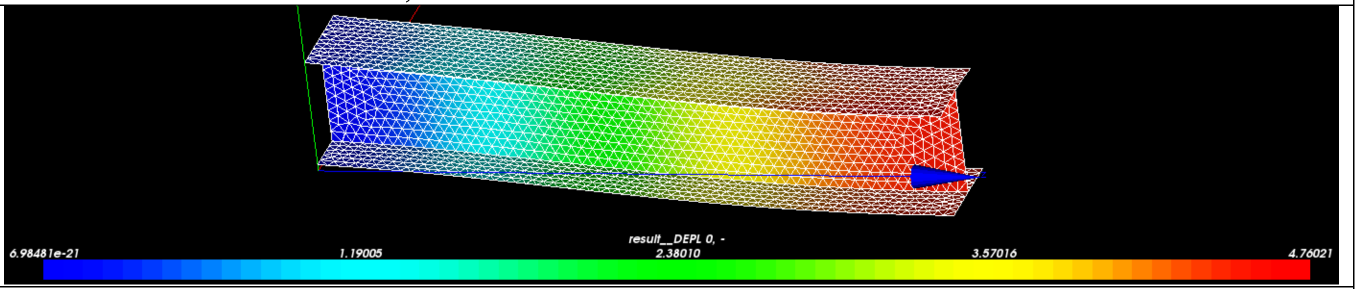
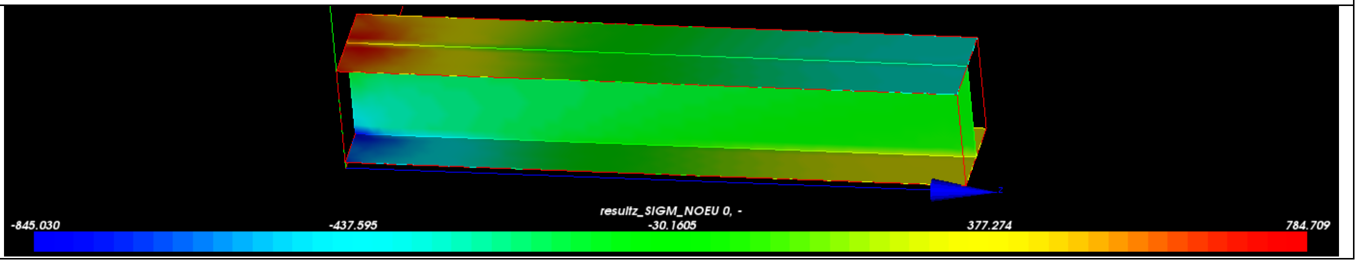
```
Surface.comm
  DEBUT :
  LIRE_MAILLAGE :              mesh
    FORMAT :                   MED
    b_format_med
  DEFI_GROUP :                 mesh
    MAILLAGE :                 mesh
    CREA_GROUP_MA :
      NOM :                    OUI
      TOUT :                   OUI
    CREA_GROUP_NO :
      b_option
        TOUT_GROUP_MA :        OUI
  MODI_MAILLAGE :              mesh
    MAILLAGE :                 mesh
    ORIE_NORM_COQUE :
      GROUP_MA :               Load
  AFFE_MODELE :                model
    MAILLAGE :                 mesh
    AFFE :
      GROUP_MA :               Guide
      PHENOMENE :              MECANIQUE
      b_mecanique
        MODELISATION :         DKT
  DEFI_MATERIAU :              steel
    ELAS :
      E :                      200000
      NU :                     0.3
  AFFE_MATERIAU :              material
    MAILLAGE :                 mesh
    AFFE :
      TOUT :                   OUI
      MATER :                  steel
  AFFE_CARA_ELEM :             element
    MODELE :                   model
    COQUE :
      COQUE_1:
        GROUP_MA :             MidThic
        EPAIS :                20
        ANGL_REP :             (1,2)
      COQUE_2:
        GROUP_MA :             TpBtThic
        EPAIS :                12.5
        ANGL_REP :             (2,3)
  AFFE_CHAR_MECA :             Suprts
    MODELE :                   model
    DDL_IMPO :
      DDL_IMPO_1:
        GROUP_NO :             Fix
        DX :                   0
        DY :                   0
        DZ :                   0
      DDL_IMPO_2:
        GROUP_NO :             Disp
        DZ :                   0
  AFFE_CHAR_MECA :             Loadz
    MODELE :                   model
    PRES_REP :
      GROUP_MA :               Load
      PRES :                   10
  MECA_STATIQUE :              result
    MODELE :                   model
    CHAM_MATER :               material
    CARA_ELEM :                element
    EXCIT :
      EXCIT_1:
        CHARGE :               Suprts
      EXCIT_2:
        CHARGE :               Loadz
  CALC_CHAMP :                 result
    RESULTAT :                 result
    TOUT_ORDRE :               OUI
    b_prec_rela
    GROUP_MA :                 Guide
    b_lineaire
      CONTRAINTE :             SIGM_ELNO
  POST_CHAMP :                 result1
    RESULTAT :                 result
    EXTR_COQUE :
      NOM_CHAM :               (SIGM_ELNO,SIEQ_ELNO)
      NUME_COUCHE :            1
      NIVE_COUCHE :            MOY
  CALC_CHAMP :                 resultz
    RESULTAT :                 result1
    TOUT_ORDRE :               OUI
    b_prec_rela
    GROUP_MA :                 Guide
    b_lineaire
      CONTRAINTE :             SIGM_NOEU
      CRITERES :               (SIEQ_ELNO,SIEQ_NOEU)
  IMPR_RESU :
    FORMAT :                   MED
    b_format_med
    b_restreint
    RESU :
      RESU_1:
        MAILLAGE :             mesh
        RESULTAT :             result
        b_info_med
        b_partie
        b_vari_el
        b_extrac
          NOM_CHAM :           (DEPL,SIEQ_NOEU)
        b_cmp
        b_med
        b_topologie
      RESU_2:
        MAILLAGE :             mesh
        RESULTAT :             resultz
        b_info_med
        b_partie
        b_vari_el
        b_extrac
          NOM_CHAM :           (SIGM_NOEU,SIEQ_NOEU)
        b_cmp
        b_med
        b_topologie
  FIN :
```

# In the Surface elements studies, like in Solid elements. That means it is not necessary to add rotational restrictions because, by applying to all the elements, of the groups assigned, DX. DY. DZ =0, you automatically force the rotations to be equal to 0.

#	Description	Figure
4.	We are now done with the command file. Go to Fishier → Enregistrer, to save the configuration we made, and close Eficas MultiCatalogue.	
5.	Activate Aster Module from the drop down list. Go to Aster → Add Study Case. Fill the gaps similar with the figure on your right. Locate your command file from your computer disk. Choose Mesh_1 as your Mesh. Change to STA11 for Aster Version Modify the Solver prameters. Finish with “Ok”	 Qt-subapplication Study case definition Name: Surface Command file: from disk — Surface/Surface/Surface.comm Mesh: from object browser — Mesh_1 ASTK services Server: localhost Aster version: STA11 Execution mode: interactif Interactive follow up ☑ Solver prameters Total memory (MB): 1024 Time (s): 1200 CPU number: 1 Save result database ☑
6.	Your Object Browser should have the same looks as the figure on the right. Right click on Solids → Run. Your analysis will start running and when it finishes Post-Pro Branch should be visible. The analysis may take a while, since we used small size of elements and our solid model is a little bit bigger than the analysis we did so far.	 Object Browser Name Geometry O OX OY OZ P1_0,6.25 P2_50,6.25 P3_100,6.25 P4_0,108.75 P5_50,108.75 P6_100 Line_1 Line_2 Line_3 Extrusion_1 Extrusion_2 Extrusion_3 Fuse_1 Fuse_2 Mesh Hypotheses Algorithms Mesh_1 Aster Surface Post-Pro Update mesh Run Status Stop Edit Copy Rename F2 Delete Del Export to ASTK Refresh F5 Expand All Find Ctrl+F

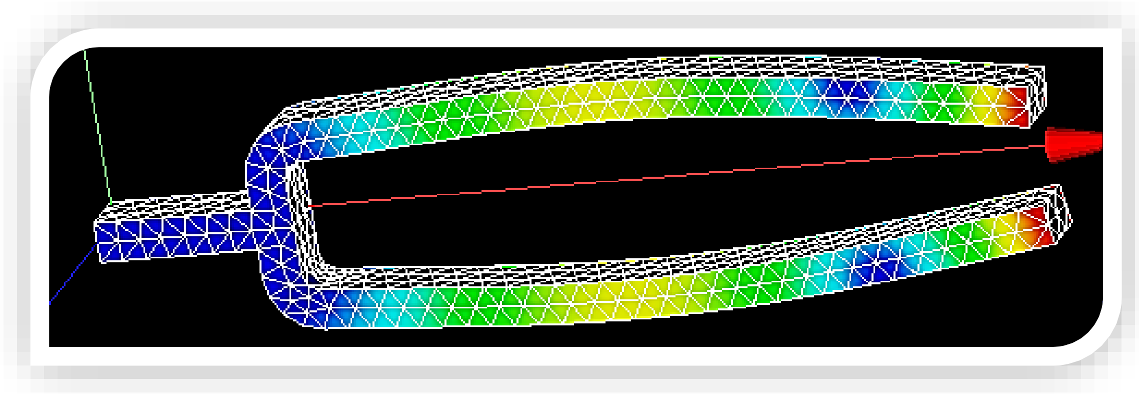
Step 11: Post Processing the Results

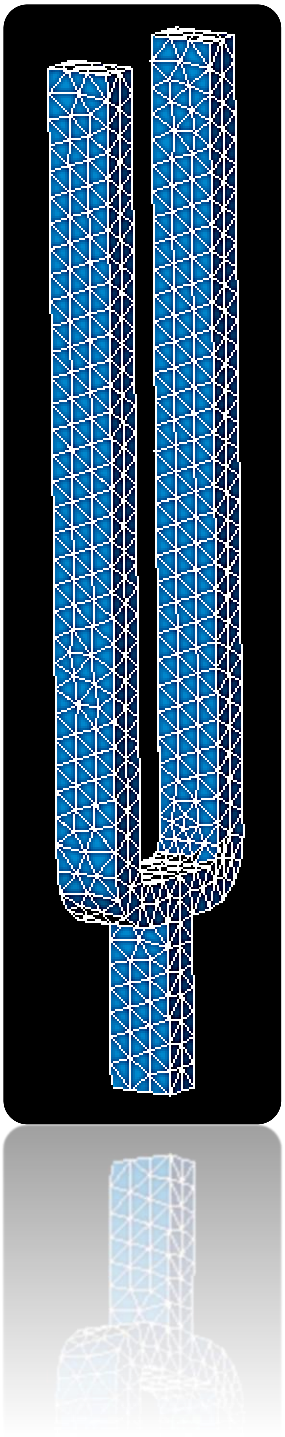
# Activate Post-Pro Module and start interrogating the results of our Finite Element Analysis.

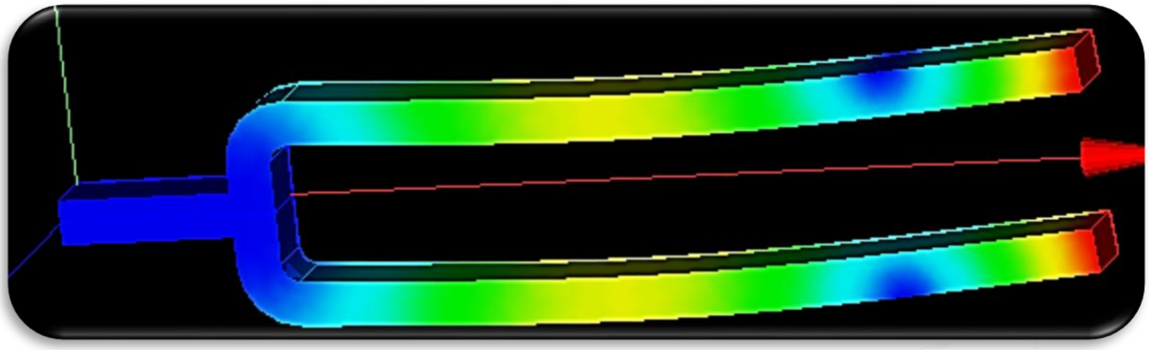
#	Description	Figure
1.	Right-click on 0,- → Deformed Shape → Tick Magnitude Coloring and change the Scale Factor to a bigger number, so you will be able to see the deformation more clear. Right-click on Def.Shape → Show Only → Representation → Surfaceframe. And you should have something similar to the figure below. Min. Deformation = 6.98e-21 mm, Max. Deformation = 4.76 mm	
2.	 result__DEPL 0, - 6.98481e-21 1.19005 2.38010 3.57016 4.76021	
3.	Same steps for the Normal Stress results. Right-click on 0,- → Scalar Map → Change Scalar Mode to SIXX. Right-click on Scalar Map → Show Only → Representation → Surface. And you should have something similar to the figure on the right. Min. Normal Stress = -845.03 MPa Max. Normal Stress = 784.7 MPa	
4.	 resultz_SIGM_NOEU 0, - -845.030 -437.595 -30.1605 377.274 784.709	

Chapter_VIII: Tunning Fork

In this next chapter we are going to get familiar with Modal Analysis.
We will study a Tuning Fork and try to acquire the most logical frequency output.







Step 1: Purpose of the FE Analysis/ Description of the Problem

In this chapter, we want to perform a Modal Analysis to investigate the natural frequencies of a Tuning Fork. The specific tuning fork is designed to tune chamber A 440Hz. In the case that the tuner does not meet the stated requirements, we will modify the geometry, material or mass of the tuner in order to get the correct frequency output.

For that reason, we will set from the start some parameters (Parametric Model) which will help us to modify the geometry's dimensions easier and without the need to sketch the tuning fork from scratch.

Step 2: Input Values for the FE Analysis

Initial Dimensions

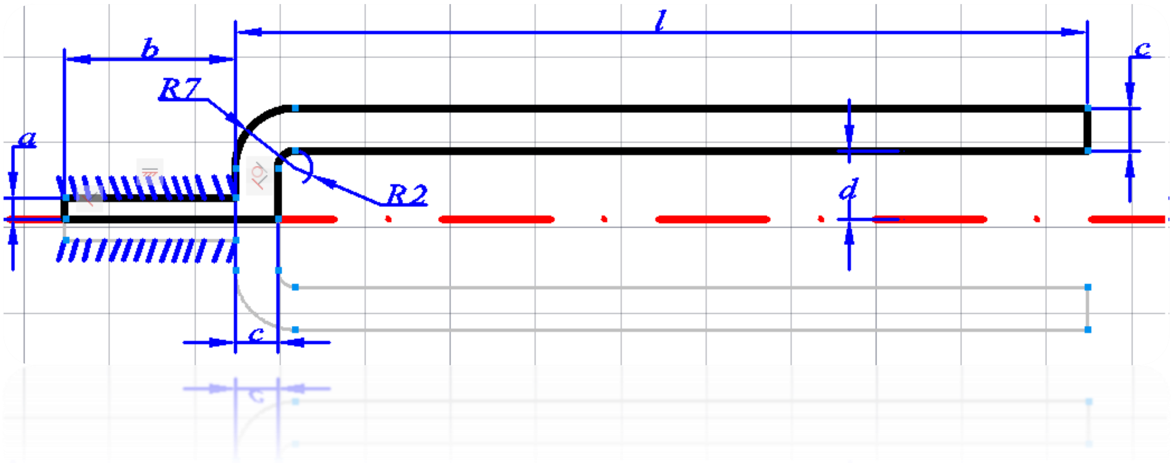
$a = 2.5$ mm;
$b = 20$ mm;
$c = 5$ mm;
$R_7 = 7$ mm;
$d = 8$ mm;
$l = 100$ mm;
$R_2 = 2$ mm;

Material

Model Material: Homogeneous, isotropic and linear elastic continuum.
Structural Steel: Young's Modulus = 200 GPa;
Poisson's Ratio = 0.3
Copper Alloy: Young's Modulus = 110 GPa;
Poisson's Ratio = 0.34

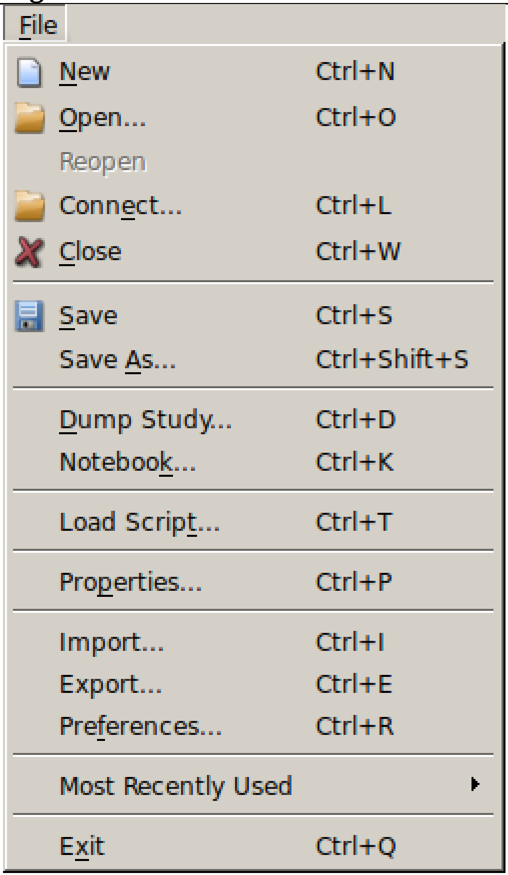
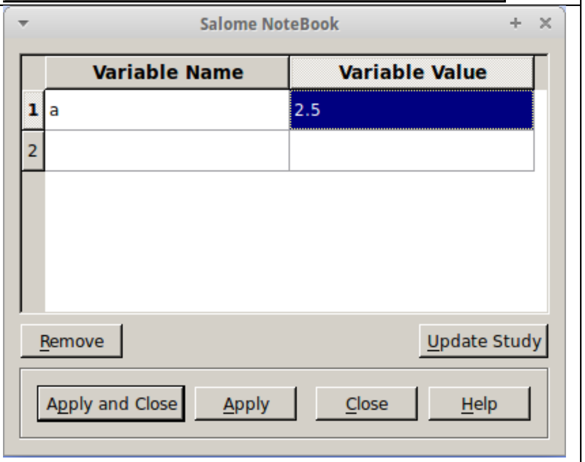
Supports

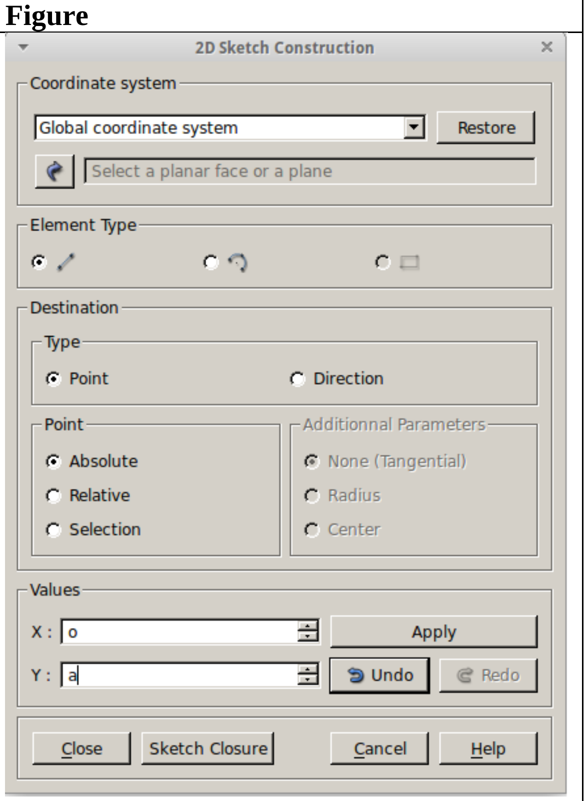
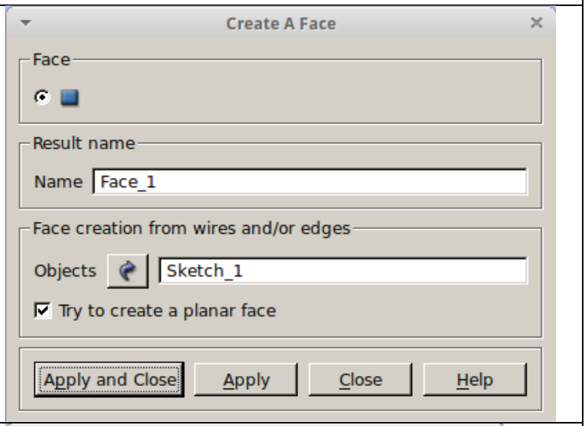
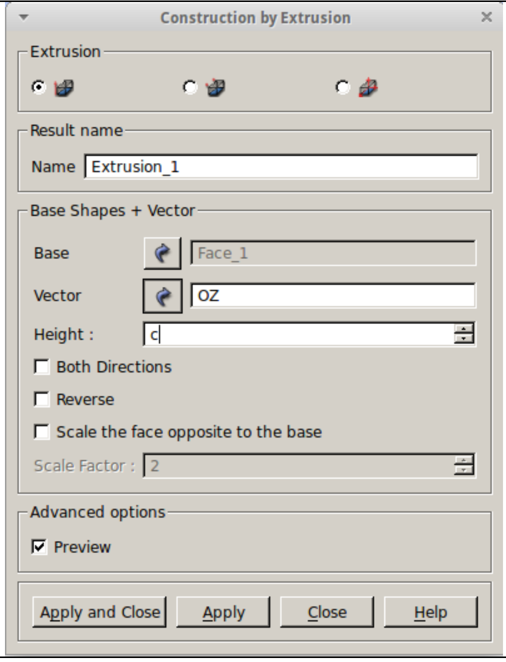
Fixed Support to "b-dimension" line (drawing will help).
Symmetric model.

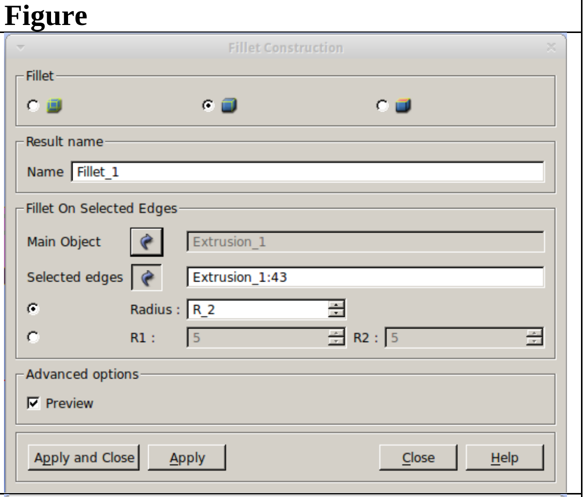
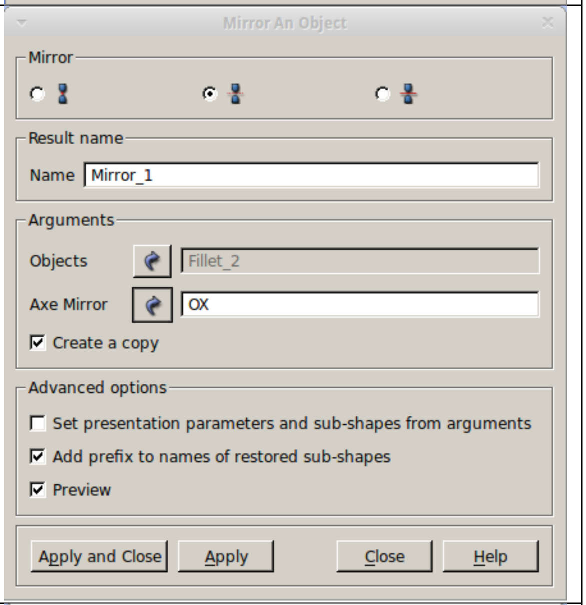
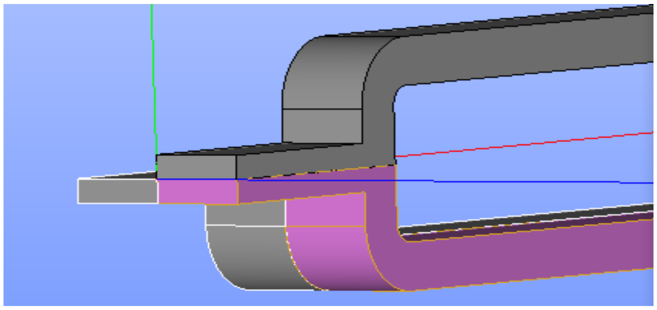
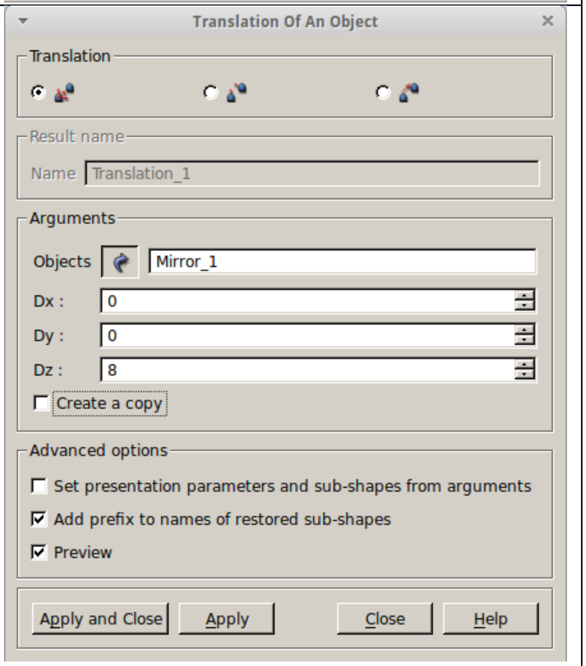


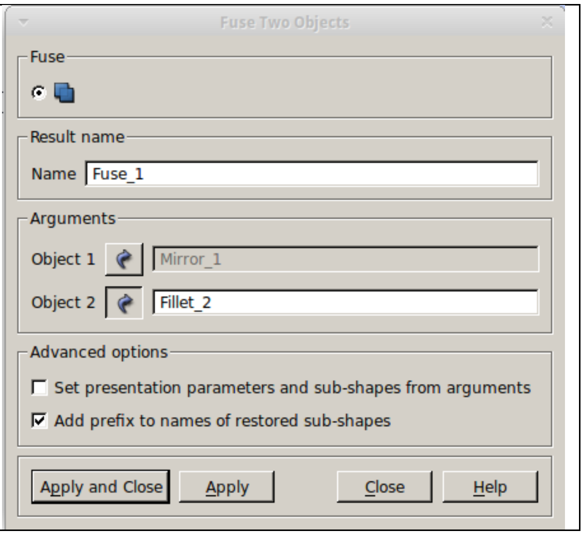
Step 3: Model Geometry

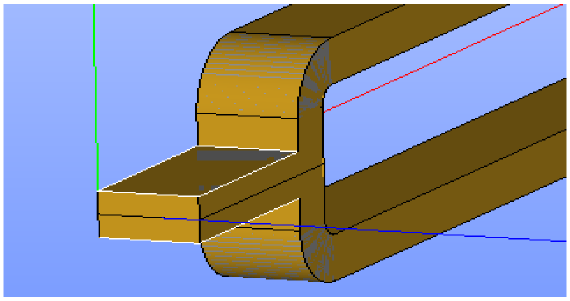
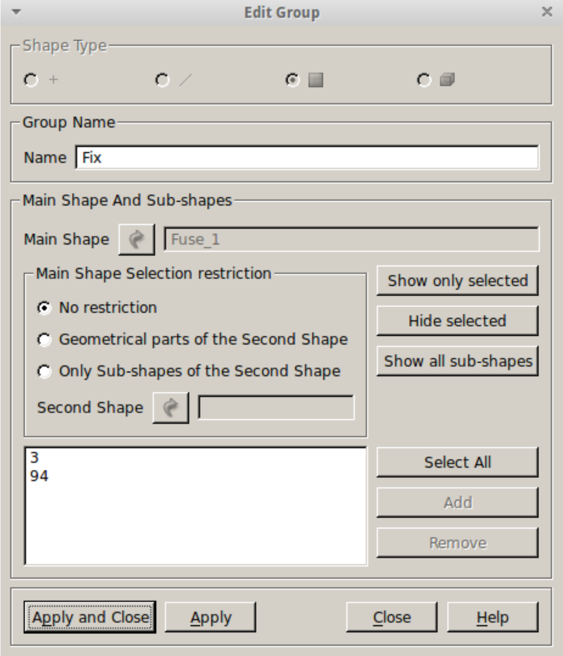
- Open up Salome_Meca and right after the GUI loads, activate Geometry Module from the drop down list.
- On this chapter we want our model to be parametric.
- There are a couple of ways to create a parametric model in Salome_Meca Platform, but because most of them are really complicated, they include coding inside, here we are going to use the most simplest one.

#	Description	Figure
1.	Having the Geometry Module activated, go to File → Notebook. Notebook is a very useful application, which allows us to input as many parameters as we desire. Notebook saves those parameters and we are able to use them in any Module we want. (Geometry, Mesh) Let's see how it works.	 File New Ctrl+N Open... Ctrl+O Reopen Connect... Ctrl+L Close Ctrl+W Save Ctrl+S Save As... Ctrl+Shift+S Dump Study... Ctrl+D Notebook... Ctrl+K Load Script... Ctrl+T Properties... Ctrl+P Import... Ctrl+I Export... Ctrl+E Preferences... Ctrl+R Most Recently Used Exit Ctrl+Q
2.	On the left column, as obvious, we arrange the Parameters Name, and on the right column we input the Parameter's Value. For our geometry I needed the following parameters: a = 2.5 o = 0 b = 5 c = 8 d = 10.5 e = 20 h = 100 R_2 = 2 R_7 = 7 Finish with "Apply" Leave Notebook open at all times, because we are going to add more parameters as we move forward with this study case.	 Salome NoteBook Remove Update Study Apply and Close Apply Close Help Variable Name: a Variable Value: 2.5
3.	Salome_Meca Geometry Module, mostly uses coordinates to sketch/ draw the models. Using Notebook for sketching is a little bit trickie because we are going to need lots of different parameters. To avoid fulling up the study with a wide range of parameters, Notebook gives us the options to use Equations to express some of the parameters.	

#	Description	Figure
4.	We have set our parameters. Have in mind that as we move forward this study, we might need to add more parameters in order to help us with the geometry/mesh. Go to New Entity → Basic → 2D Sketch. As you already know for the construction of a 2D sketch, we were pointing out each point's coordinates (x, y). Now instead of arranging numbers for Values, we will input the parameters we set before. For example: The first point, which originally would have been (0, 0) now according to our parameters will be (o, o). For this geometry we will need 3 more parameters that we haven't set it up before. f = a + d (13) j = h + e (120) g = b + e (25) Next points: (o, o) (o, a) (e, a) (e, f) (j, f) (j, c) (g, c) (g, o) Finish with "Sketch Closure"	 2D Sketch Construction Coordinate system: Global coordinate system Element Type: ● Destination Type: ● Point ○ Direction Point: ● Absolute ○ Relative ○ Selection Additionnal Parameters: ● None (Tangential) ○ Radius ○ Center Values X : o Y : a
5.	In order to create the whole model, first we need to create a face of the 2D sketch. Go to New Entity → Build → Face. Finish with "Apply and Close"	 Create A Face Result name: Name Face_1 Face creation from wires and/or edges: Objects Sketch_1 ☑ Try to create a planar face
6.	Go to New Entity → Generation → Extrusion. Fill in the gaps according to the figure on the right. Height: c (c=8mm). Finish with "Apply and Close"	 Construction by Extrusion Result name: Name Extrusion_1 Base: Face_1 Vector: OZ Height : c ☐ Both Directions ☐ Reverse ☐ Scale the face opposite to the base Scale Factor : 2 Advanced options: ☑ Preview

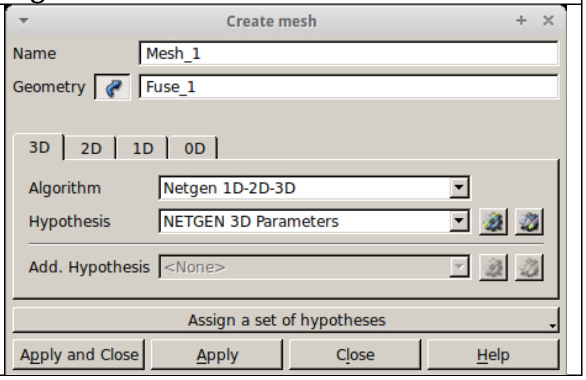
#	Description	Figure
	Now we need to create our fillets. Go to Operations → Fillet 3D. Choose the 2nd Algorithm which will allow us to select edges, and input the same values as shown in the figure on the right. Finish with "Apply and Close" Do the same for the 7mm fillet, but be careful to choose the first fillet as "Main Object". Finish with "Apply and Close"	 Fillet Construction Fillet: ○ ● ○ Result name Name: Fillet_1 Fillet On Selected Edges Main Object: Extrusion_1 Selected edges: Extrusion_1:43 ● Radius : R_2 ○ R1 : 5 R2 : 5 Advanced options ☑ Preview
7.	Now it is time to create the mirror image. Go to Operations → Transformation → Mirror Image. Input the following values as shown in the figure on the right. Finish with "Apply and Close"	 Mirror An Object Mirror: ○ ● ○ Result name Name: Mirror_1 Arguments Objects: Fillet_2 Axe Mirror: OX ☑ Create a copy Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes ☑ Preview
8.	Mirror image was created a little bit away from the location we wanted it to be. Go to Operations → Transformation → Translation. We want our mirror image to move 8mm to the OZ direction. Make sure you uncheck the "Create a Copy" Finish with "Apply and Close" 	 Translation Of An Object Translation: ● ○ ○ Result name Name: Translation_1 Arguments Objects: Mirror_1 Dx : 0 Dy : 0 Dz : 8 ☐ Create a copy Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes ☑ Preview

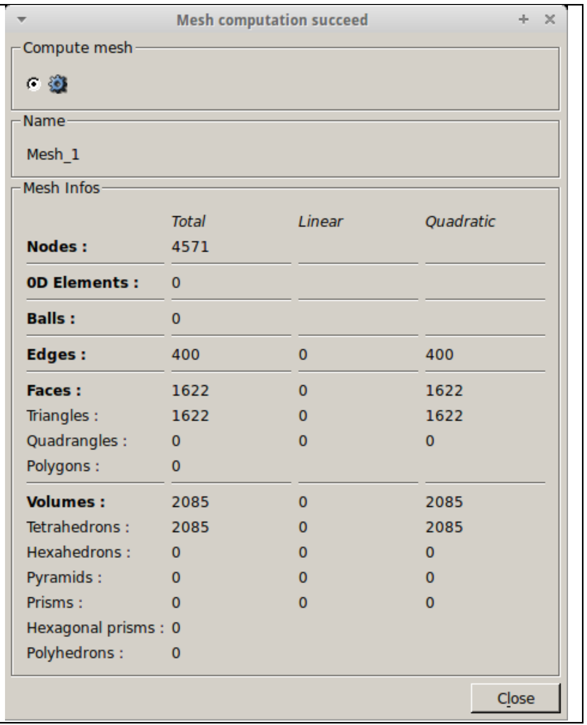
9.	Last step before arranging our groups, will be to couple the extrusion with the mirror image. Go to Operations → Boolean → Fuse. And input the following values as shown in the figure on the right. Finish with “Apply and Close”	 Fuse Two Objects Fuse Result name Name: Fuse_1 Arguments Object 1: Mirror_1 Object 2: Fillet_2 Advanced options ☐ Set presentation parameters and sub-shapes from arguments ☑ Add prefix to names of restored sub-shapes

#	Description	Figure
10.	We are going to need only two different groups for this study case. Our Fixed Support which would be in the left top and bottom faces.  And a Guide group which will contain all the faces we created. Finish with "Apply and Close"	 Edit Group Shape Type: ● Group Name Name: Fix Main Shape: Fuse_1 Main Shape Selection restriction ● No restriction ○ Geometrical parts of the Second Shape ○ Only Sub-shapes of the Second Shape 3 94 Apply and Close
11.	This concludes our Geometry creation session. Save in a proper location with a proper name.	

Step 4: Meshing Geometry

# After finishing from the Geometry creation, go to the drop down list and activate Mesh Module.

#	Description	Figure
1.	Meshing our model. Go to Mesh → Create Mesh. Input for Algorithm: NETGEN 1D-2D-3D Input for Hypothesis: NETGEN 3D Parameters, with maximum and minimum size of elements 3 and 2. Finish with "Apply and Close"	 Create mesh Name: Mesh_1 Geometry: Fuse_1 3D Algorithm: Netgen 1D-2D-3D Hypothesis: NETGEN 3D Parameters Add. Hypothesis: <None>

2.	Right-click on Mesh_1 → Compute. Right-click on Mesh_1 → Convert to/from quadratic. Right-click on Mesh_1 → Compute. Mesh information shown on the right figure.	

Mesh computation succeed

Compute mesh

Name

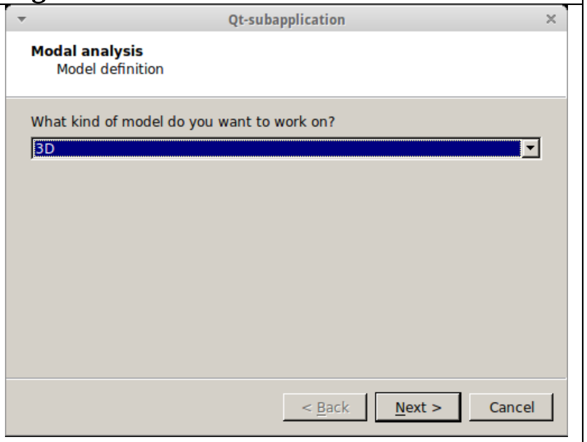
Mesh_1

Mesh Infos

	Total	Linear	Quadratic
Nodes :	4571		
0D Elements :	0		
Balls :	0		
Edges :	400	0	400
Faces :	1622	0	1622
Triangles :	1622	0	1622
Quadrangles :	0	0	0
Polygons :	0		
Volumes :	2085	0	2085
Tetrahedrons :	2085	0	2085
Hexahedrons :	0	0	0
Pyramids :	0	0	0
Prisms :	0	0	0
Hexagonal prisms :	0		
Polyhedrons :	0		

Step 5, 6, 7, 8, 9, 10: Salome_Meca Modal Analysis Wizard

# After completing our geometry, arranging the correct groups, using the most suitable algorithm and hypothesis to express our mesh, now its time to make some more modification for our modal analysis.

#	Description	Figure
1.	Select Aster Module from the drop down list. Go to Aster → Wizards → Modal Analysis. First, define the type of model. Click "Next". Input Mesh_1 for mesh. Click "Next". In the next window insert Young's Modulus = 2e+11, Density = 7850 (structural steel) and Poisson's Ratio = 0.3. Click "Next". Fix Group (DX=0, DY=0, DZ=0). Click "Next" Set Number of Modes (N) = 8. Save to a proper location. Finish with "Finish"	

Qt-subapplication

Modal analysis

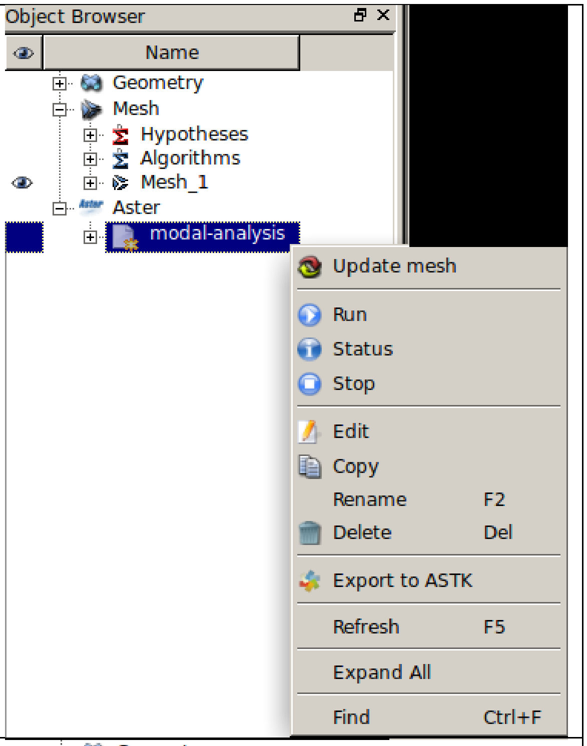
Model definition

What kind of model do you want to work on?

3D

2. Our Modal analysis will appear in the Aster branch. Right-click on it and choose “Run”.

After Code_Aster runs the modal analysis, and if everything are set correctly, Post-Pro branch will appear, containing all the information we need.



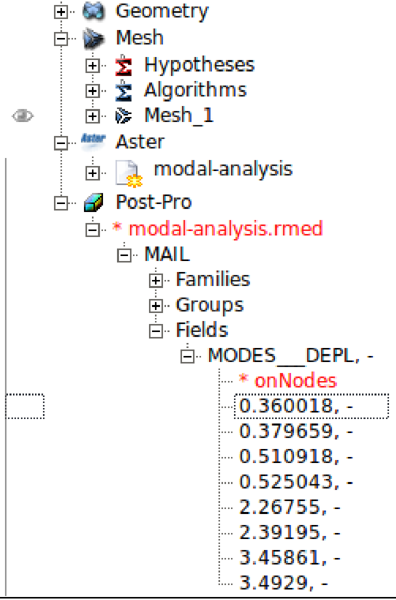
3. When Code_Aster finish running the modal analysis, Post-Pro branch will appear.

In the “Fields” area, there are the 8 different modes (which we selected before) showing the different frequencies.

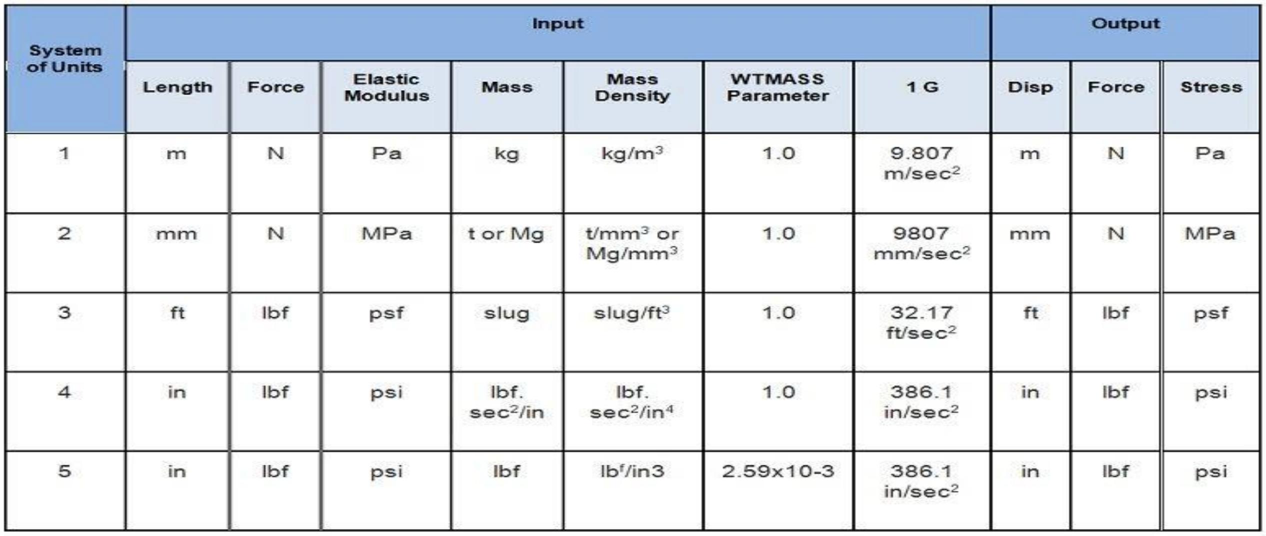
You can easily notice that the frequency outputs are smaller than expected.

That is because we used millimeters when we created the geometry and the rest are in basic SI Units. There is a consistency table of units that we have to take into consideration every time we perform a complete Finite Element Analysis, which is shown below.

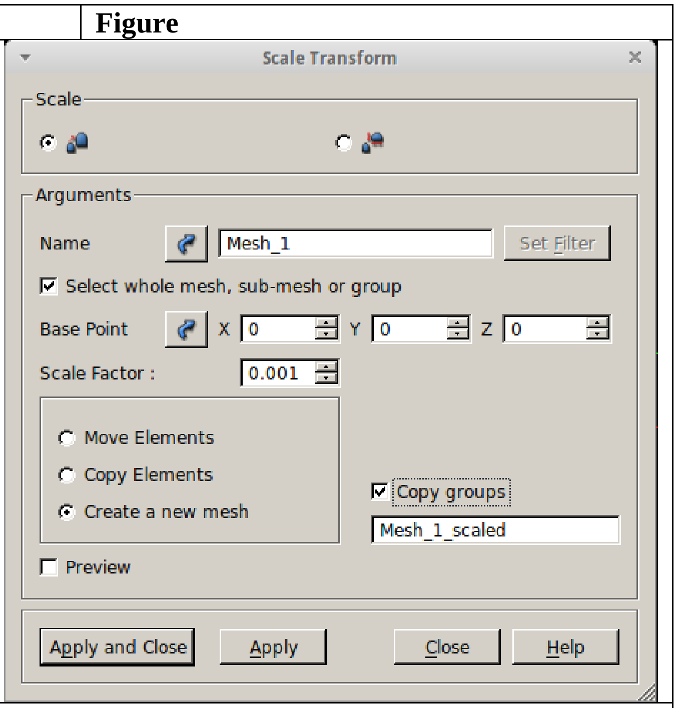
All we need to do now is to change our geometry to meters.



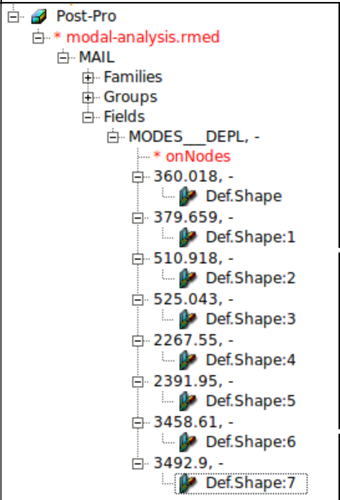
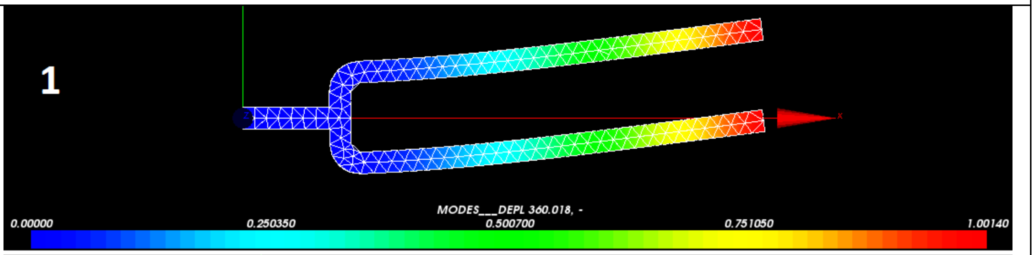
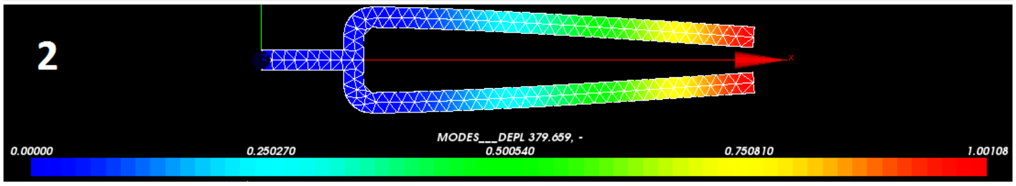
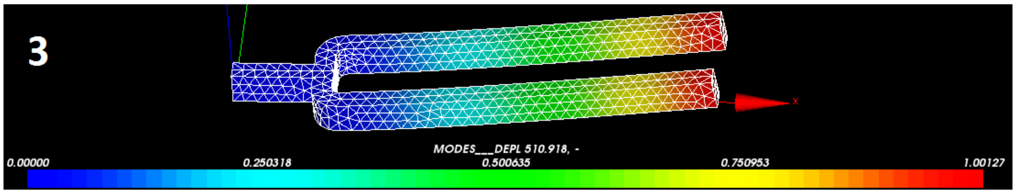
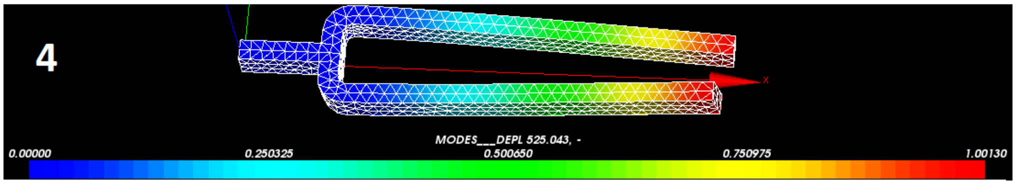
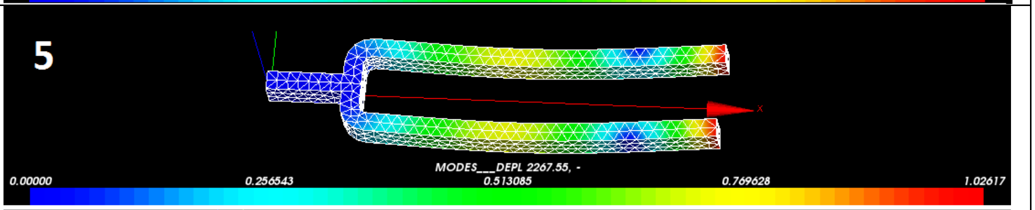
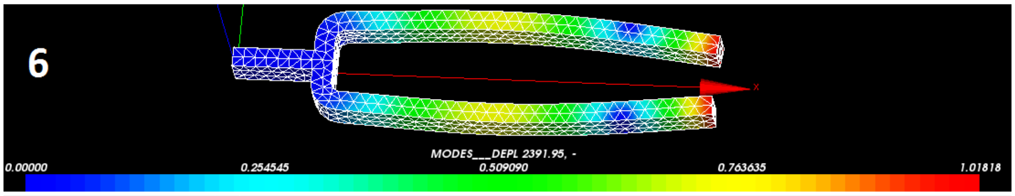
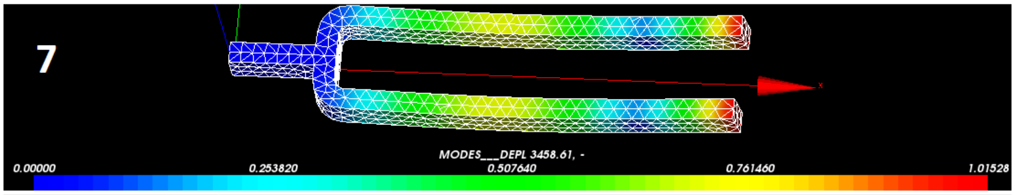
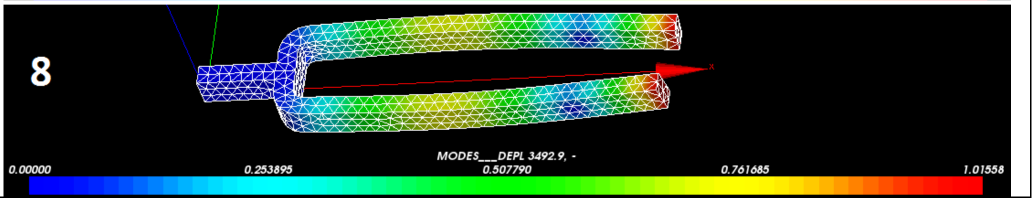
4.



System of Units	Input							Output		
	Length	Force	Elastic Modulus	Mass	Mass Density	WTMASS Parameter	1 G	Disp	Force	Stress
1	m	N	Pa	kg	kg/m^3	1.0	9.807 m/sec^2	m	N	Pa
2	mm	N	MPa	t or Mg	t/mm^3 or Mg/mm^3	1.0	9807 mm/sec^2	mm	N	MPa
3	ft	lbf	psf	slug	slug/ft^3	1.0	32.17 ft/sec^2	ft	lbf	psf
4	in	lbf	psi	lbf. sec^2/in	lbf. sec^2/in^4	1.0	386.1 in/sec^2	in	lbf	psi
5	in	lbf	psi	lbf	lb^f/in3	2.59x10-3	386.1 in/sec^2	in	lbf	psi

#	Description	Figure
5.	To change our geometry units from milimeters to meters, all we have to do is duplicate our mesh and use a scale factor to adjust the units. Activate Mesh Module. Go to Modification → Transformation → Scale Transform. First click the "Select whole mesh, sub-mesh or group" option. Then fill in the gaps and choose the correct options as shown in the figure on the right. Finish with "Apply and Close"	 Scale Transform Scale Arguments Set Filter Apply and Close Apply Close Help Name: Mesh_1 ☑ Select whole mesh, sub-mesh or group Base Point X 0 Y 0 Z 0 Scale Factor : 0.001 ○ Move Elements ○ Copy Elements ● Create a new mesh ☑ Copy groups Mesh_1_scaled ☐ Preview
6.	Right click on the new mesh, "Mesh_1_scaled" and compute it. You will notice the the mesh information is not like the others. The thing we did here was, to copy our previous mesh with the previous Algorithms and Hypothesis. The only difference is that with the help of the Scale Transformation of 0,001 we converted the milimeters to meters.	
7.	Activate Aster Module, right click on the modal analysis and choose "Edit". You just need to change the "Mesh_1" to the new scaled mesh, "Mesh_1_scaled". Click "Ok", right click again and "Run" the analysis.	

Step 11: Post Processing of the Results

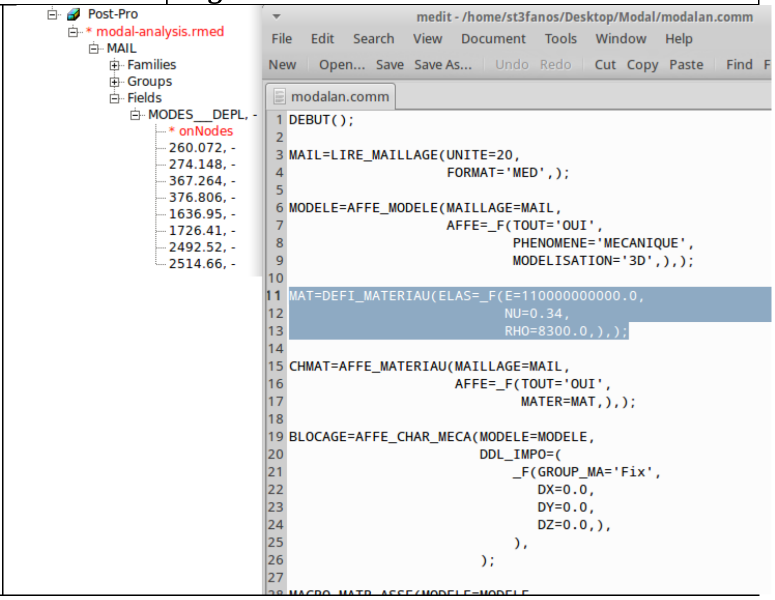
#	Description	Figure
1.	Now that we have the correct units and the frequency outputs seem normal, we can add deformed shape to each one of the frequencies to see how much will the deformation be and also in what way the tuning fork is going to be deformed.	
2.	 Post-Pro * modal-analysis.rmed MAIL Families Groups Fields MODES___DEPL, - * onNodes 360.018, - Def.Shape 379.659, - Def.Shape:1 510.918, - Def.Shape:2 525.043, - Def.Shape:3 2267.55, - Def.Shape:4 2391.95, - Def.Shape:5 3458.61, - Def.Shape:6 3492.9, - Def.Shape:7	 1 0.00000 0.250350 MODES___DEPL 360.018, - 0.500700 0.751050 1.00140  2 0.00000 0.250270 MODES___DEPL 379.659, - 0.500540 0.750810 1.00108  3 0.00000 0.250318 MODES___DEPL 510.918, - 0.500635 0.750953 1.00127  4 0.00000 0.250325 MODES___DEPL 525.043, - 0.500650 0.750975 1.00130
3.		 5 0.00000 0.256543 MODES___DEPL 2267.55, - 0.513085 0.769628 1.02617  6 0.00000 0.254545 MODES___DEPL 2391.95, - 0.509090 0.763635 1.01818  7 0.00000 0.253820 MODES___DEPL 3458.61, - 0.507640 0.761460 1.01528  8 0.00000 0.253895 MODES___DEPL 3492.9, - 0.507790 0.761685 1.01558

Step $: Modify Model

i. Changing the Material

- # The main purpose of this Modal Analysis, is to get familiar with this kind of analysis and also try to adjust our model in order to give us a frequency output of 440 Hz.
- # In order to do that we are going to make some adjustments to our model and see how it reacts.
- # First we are going to change the material of our tuning fork.
- # To do that we will need to change the Young's Modulus, the Poison's Ration and the Density inputs.
- # We are interested in using Copper Alloy instead of Structural Steel.
- # Copper Alloy's material properties:

Copper Alloy's material properties:	Young's Modulus =	1.1e11 Pa
	Poison's Ratio =	0.34
	Density =	8300 kg/m^3

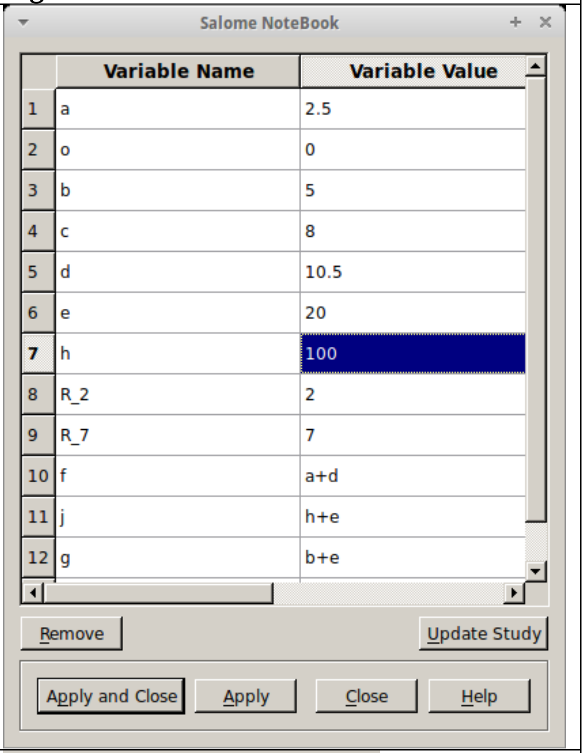
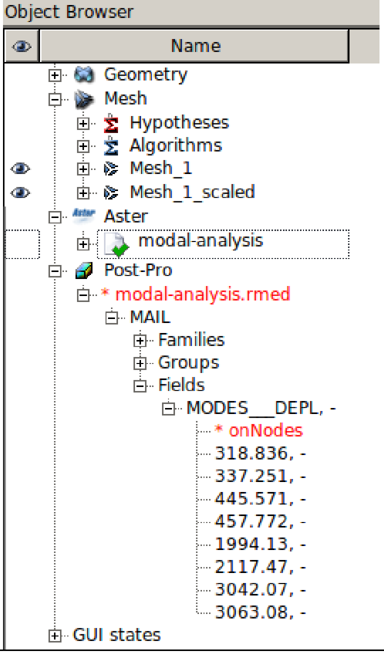
#	Description	Figure
1.	Open your "yourname".comm file with your favourite text editor. Go to DEFI_MATERIAL lines and change the inputs according to the new material's properties. Save your changes and head back to Salome_Meca. Activate Aster Module, and re-run the Analysis.	

```
DEBUT();

MAIL=LIRE_MAILLAGE(UNITE=20,
                   FORMAT='MED',);

MODELE=AFFE_MODELE(MAILLAGE=MAIL,
                   AFFE=_F(TOUT='OUI',
                           PHENOMENE='MECANIQUE',
                           MODELISATION='3D',),);

MAT=DEFI_MATERIAU(ELAS=_F(E=110000000000.0,
                          NU=0.34,
                          RHO=8300.0,),);

CHMAT=AFFE_MATERIAU(MAILLAGE=MAIL,
                    AFFE=_F(TOUT='OUI',
                            MATER=MAT,),);

BLOCAGE=AFFE_CHAR_MECA(MODELE=MODELE,
                       DDL_IMPO=(
                           _F(GROUP_MA='Fix',
                              DX=0.0,
                              DY=0.0,
                              DZ=0.0,),
                           ),
                       );
```

- # When the Post-Pro branch appears again, you will notice that the frequency outputs are changed, but instead of getting closer to 440Hz, they went smaller.

ii. Changing the Dimensions.

- Since, changing the material did not help us get the required frequency results, we will try now and change some dimensions from our tuner and see how it affects the frequency results. Changing all the dimension would not be much of a help for us, we will modify the l=length dimension and possibly the thickness of the tuner.
- This is why we assigned the dimensions into Parameters (Notebook) in the beginning of our task.
- Now we just have to open up Notebook and change the parameter inputs and just re-run the analysis.
- Let’s see how we are going to achieve that.

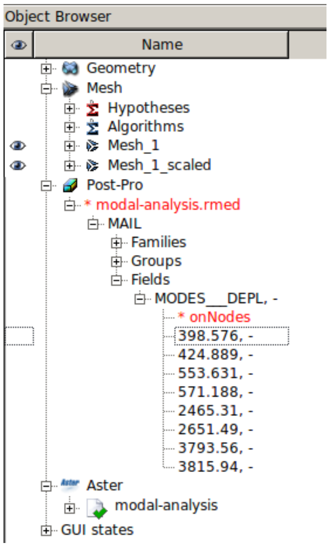
#	**Description**	**Figure**
1.	Go to File → Notebook Change the “h” Parameter, which is the length of the fork, from 100mm to 90mm. Click on “Update Study” Then the whole UI of Salome_Meca is going to change and in the Object Browser there will only be Geometry and Mesh branches.	Salome NoteBook Variable Name / Variable Value 1 a 2.5 2 o 0 3 b 5 4 c 8 5 d 10.5 6 e 20 7 h 100 8 R_2 2 9 R_7 7 10 f a+d 11 j h+e 12 g b+e Remove Update Study Apply and Close Apply Close Help
2.	Unfortunately we will need to re-compute the mesh, create a new modal analysis wizard and run the analysis in order to see the new frequency results. The steps you need to do now are: * Right click on Mesh_1 and choose compute, right click again and convert to quadratic, compute again. * Right click on Mesh_1_scaled and choose compute, right click again and convert to quadratic, compute again. * Activate Aster Module. * Go to Aster → Wizards → Modal Analysis. * Input the correct data and save to a proper location. * Remember that we will use Copper Alloy as material here. * Right click on the new modal-analysis and Run it. The figure on the right shows my own results.	Object Browser Name Geometry Mesh Hypotheses Algorithms Mesh_1 Mesh_1_scaled Aster modal-analysis Post-Pro * modal-analysis.rmed MAIL Families Groups Fields MODES___DEPL, - * onNodes 318.836, - 337.251, - 445.571, - 457.772, - 1994.13, - 2117.47, - 3042.07, - 3063.08, - GUI states

# You can see that the frequency outputs are a little bit bigger than the previous ones. Which means that as the length of the fork gets smaller the frequency gets higher.

# In order to hit 440Hz, we will need to make the fork's length a little smaller.

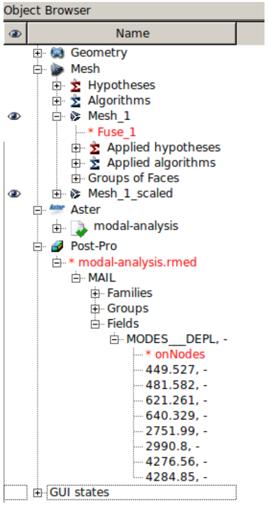
# Open Notebook again and change the length parameter from 90mm to 80mm.

# After re-computing everything with the length set to 80mm, my results are:



```
Object Browser
Name
⊞ Geometry
⊟ Mesh
    ⊞ Hypotheses
    ⊞ Algorithms
    ⊞ Mesh_1
    ⊞ Mesh_1_scaled
⊟ Post-Pro
    ⊟ * modal-analysis.rmed
        ⊟ MAIL
            ⊞ Families
            ⊞ Groups
            ⊟ Fields
                ⊟ MODES___DEPL, -
                    * onNodes
                    398.576, -
                    424.889, -
                    553.631, -
                    571.188, -
                    2465.31, -
                    2651.49, -
                    3793.56, -
                    3815.94, -
⊟ Aster
    ⊞ modal-analysis
⊞ GUI states
```

# We will give it one more try to 75mm and see if we get closer to 440Hz. Here are my final results:



```
Object Browser
Name
⊞ Geometry
⊟ Mesh
    ⊞ Hypotheses
    ⊞ Algorithms
    ⊟ Mesh_1
        * Fuse_1
        ⊞ Applied hypotheses
        ⊞ Applied algorithms
        ⊞ Groups of Faces
    ⊞ Mesh_1_scaled
⊟ Aster
    ⊞ modal-analysis
⊟ Post-Pro
    ⊟ * modal-analysis.rmed
        ⊟ MAIL
            ⊞ Families
            ⊞ Groups
            ⊟ Fields
                ⊟ MODES___DEPL, -
                    * onNodes
                    449.527, -
                    481.582, -
                    621.261, -
                    640.329, -
                    2751.99, -
                    2990.8, -
                    4276.56, -
                    4284.85, -
⊞ GUI states
```